Canada - Ontario Mineral Development Agreement

Entente Canada - Ontario sur l'exploration minérale

Canada

Ontario

CANADA - ONTARIO

MINERAL DEVELOPMENT AGREEMENT

SIMULATION OF BULK MINING AT DEPTH WITH BACKFILL IN ONTARIO MINES

Final Report - Volume 5

MINE2D USER'S MANUAL

by

Mines Research Department
INCO Limited

65 pp

Work on the project was conducted under the auspices of Canada Centre for Mineral and Energy Technology, Energy, Mines and Resources Canada under the Canada/Ontario Mineral Development Agreement.

DSS File No. 14SQ.23440-6-9033
DSS Contract Serial No. 23440-6-9033/01-SQ
CANMET Project No. 6-9033
CANMET Scientific Authority: Y.S. Yu

MICROMEDIA

93-01527/

ABSTRACT

MINE2D is a two-dimensional boundary element program which calculates stresses, strains, displacements and failure zones resulting from the simulation of bulk mining in massive orebodies. The program includes a plasticity formulation to simulate localized failure regions around mined openings, a multiple material formulation to model regions of different materials such as backfill, and a displacement discontinuity formulation to model features such as faults.

MINE2D consists of two separate computer codes: an analysis program MINE, and a graphical processor OUTPUT. This volume is the User's Manual for MINE2D and explains the operation of MINE and OUTPUT. An example analysis is presented to assist the user.

SOMMAIRE

MINE2D est un programme d'éléments frontières bi-dimensionnel qui calcule les contraintes, les tensions, les déplacements et les zones de rupture résultant de la simulation du minage en vrac des gisements massifs. Le programme tient compte des propriétés plastiques des matériaux pour simuler les zones de rupture locale autour des excavations minières. Le logiciel permet l'introduction de matériaux aux propriétés différentes, tel le remblai à l'intérieur du gisement. Enfin, la discontinuité des déplacements, lors de la modélisation des failles par exemple, est introduite dans le programme.

MINE2D consiste en deux (2) modules séparés: un programme d'analyse MINE et un processeur graphique OUTPUT. Cet ouvrage est le guide de l'usager pour le programme MINE2D et explique le fonctionnement des modules MINE et OUTPUT. Un exemple d'analyse est présenté pour assister l'usager.

PREFACE

This project entitled "Simulation of Bulk Mining at Depth in Ontario Mines with Backfill" consists of two major components: the development of a numerical model, and field verification of the model. The project, which began in 1986, was expected to take four years ending in September 1990 but was extended until March 1991. This report is the fifth of six volumes which describe the results of the project.

The work was initiated by Inco Limited and is partially funded through the MDA Program (joint Canada-Ontario Mineral Development Agreement, DSS file 14SQ.23440-6-9033). The project is administered by Canmet and their assistance is greatly appreciated.

LIMITATION OF LIABILITY

Canmet, Inco Limited, and their subcontractors assume no liability whatsoever with respect to any use of MINE2D or any portion thereof or with respect to any damages or losses which may result from such use, including (without limitation) loss of time, money or goodwill which may arise from the use of MINE2D. In no event shall Canmet, Inco Limited, or their subcontractors be responsible for any indirect, special, incidental or consequential damages arising from the use of MINE2D.

TABLE OF CONTENTS

		Page
1.0	INTRODUCTION	1-1
2.0	THE BOUNDARY ELEMENT METHOD IN MINE2D	2-1
	2.1 Boundary Element Formulation	2-1
	2.2 Modelling Plasticity	2-1
	2.3 Multiple Materials	2-1
	2.4 Sequential Mining	2-2
	2.5 Fault Slip - Seam Mining Simulation	2-2
	2.6 Solution Process	2-3
3.0	NUMERICAL MODELLING WITH MINE2D	3-1
	3.1 Software Overview	3-1
	3.2 Program Requirements and Limitations	3-1
	3.3 Modelling Sequence	3-3
	3.4 Energy Calculation	3-4
	3.5 Sign Convention and Units	3-5
	3.6 Hardcopy Plots	3-5
4.0	INPUT PREPARATION	4-1
	4.1 Planning the Model	4-1
	4.2 Preparing the Input Data File	4-1
	4.2.1 Job Description	4-3
	4.2.2 Control Parameters	4-3
	4.2.3 Zone Geometry Definition	4-4
	4.2.4 Coordinate List	4-7
	4.2.5 Stress States	4-7
	4.2.6 Material Properties - Failure Criteria	4-8
	4.2.7 Grid Definition	4-11
	4.2.8 Mining Steps	4-11
	4.2.9 Restarting Result Calculation	4-13
5.0	MINE2D ANALYSIS PROGRAM (MINE)	5-1
	5.1 Running the Analysis	5-1
	5.2 Convergence Issues	5-3
	5.2.1 Coefficient Matrix Solution	5-3
	5.2.2 Load Step Solution	5-4
	5.3 Result Files	5-5
6.0	MINE2D OUTPUT PROGRAM (OUTPUT)	6-1
	6.1 Menu Operation	6-1
	6.2 Parameter Editing	6-2
	6.3 Display Control	6-2
	6.4 Calculated Data Plots	6-3
	6.5 Function Keys	6-4
7.0	EXAMPLE PROBLEM	7-1
8.0	REFERENCES	8-1

Table of Contents
(continued)

Page

APPENDIX I - SAMPLE.INP FILE LISTING I-1

APPENDIX II - SOFTWARE INSTALLATION NOTES II-1

LIST OF FIGURES

		page
Figure 4-1	MINE2D Input Data File Format	4-2
Figure 4-2	Example Mining Step to Restart MINE2D	4-13
Figure 5-1	Starting MINE Analysis Program	5-1
Figure 5-2	MINE Analysis Monitoring Display	5-2
Figure 5-3	Model.OUT File	5-6
Figure 7-1	Layout for EXAMPLE1 Model	7-2
Figure 7-2	Initial Screen Display of the Main Menu in OUTPUT	7-6
Figure 7-3	Using VIEW INPUT FILE to Check Mining Step 5	7-6
Figure 7-4	Editing the Plot COORDINATE RANGE Parameters	7-7
Figure 7-5	Mining Step Redisplayed to Show Selected Area	7-7
Figure 7-6	Sigma:1 Stress Contour Plot with Sigma:n on Fault	7-8
Figure 7-7	Principal Stress Axes Orientation Plot	7-8
Figure 7-8	Editing the STRESS RANGE Parameters	7-9
Figure 7-9	Sigma:1 Contoured to Show Stress in Backfill	7-9
Figure 7-10	Sigma:2 Stress Contour Plot with Sigma:s on Fault	7-10
Figure 7-11	Stress in the X-Axis Direction	7-10
Figure 7-12	Safety Factor Contour Plot	7-11
Figure 7-13	Failure Zone Plot of Plastic Yielding Areas	7-11
Figure 7-14	Total Displacement Contour Plot	7-12
Figure 7-15	Displacement Orientation Plot	7-12
Figure 7-16	Displacement in the X-Axis Direction	7-13
Figure 7-17	Displacement in the X-Axis Direction	7-13
Figure 7-18	Eps:1 Plastic Strain Plot with Eps:n on Fault	7-14
Figure 7-19	Stored Elastic Strain Energy Density Plot	7-14
Figure 7-20	Sigma:1/n Stress Plot after Mining Pillar	7-15
Figure 7-21	Total Displacement Plot after Mining Pillar	7-15

LIST OF TABLES

		page
Table 3-1	Memory Requirements and Problem Size for MINE	3-2
Table 3-2	System of Units	3-5
Table 6-1	OUTPUT Program Options	6-1
Table 6-2	Calculated Data Plots Available in OUTPUT	6-3

1.0 INTRODUCTION

MINE2D is a two-dimensional boundary element computer program for calculating stresses, displacements, strains, safety factor, failure zones, and energy change around underground mine openings. This code is designed to simulate bulk mining with backfill, and contains the following features to support this capability.

The code has both linearly-elastic surface elements and plastic volume elements to simulate localized zones of plastic yield within an elastic continuum.

Faults are simulated using displacement discontinuity elements.

Multiple materials may be simulated by assigning different material properties to zones of volume elements.

Backfill may be placed in zones of volume elements to permit the simulation of backfill response to deformation.

Model geometry is constructed by defining linear, quadrilateral and triangular "zones" of elements.

Multiple mining steps and restart capability are provided to model mining and backfill sequences.

The mathematical formulation and features of MINE2D are described in detail in "Simulation of Bulk Mining at Depth with Backfill in Ontario Mines, Volume 2 - Mathematical Formulations and Code Development". Section 2 of this manual summarizes these features.

MINE2D consists of two separate programs: MINE, the main analysis program, and OUTPUT, a graphical processor. The input data file is prepared in the required format by using any ASCII file editor. The input file may be checked by using OUTPUT to display the geometry of each mining step. MINE is then run to calculate the results. The menu-driven OUTPUT program is run to display the analysis results. Details of MINE2D operation are discussed in Sections 3 through 6. An example application of MINE2D is provided in Section 7.

Plane Strain Limitation

The formulation for MINE2D assumes a two-dimensional plane strain state. Plane strain is associated with long excavations of constant cross-section through the model and acted on by loads in the plane of the cross-section. While this restricts the mining problems that can be accurately modelled, MINE2D may be useful to compare alternate plans where non-elastic behaviour is important.

2.0 THE BOUNDARY ELEMENT METHOD IN MINE2D

2.1 Boundary Element Formulation

MINE2D is based upon the indirect boundary element method. The theory behind this method is described in detail by Banerjee and Butterfield (1981). This method uses "fictitious force" elements to define the excavation surfaces.

Linear-strength elements are used rather than constant-strength elements. This means that nodes are located at corners of elements rather than at element centres. As a result, better accuracy can be obtained than with constant-strength elements.

The formulation is described in detail in Volume 2.

2.2 Modelling Plasticity

When plasticity is introduced into the method, the term boundary element becomes something of a misnomer. Zones of quadrilateral shaped (initial stress) volume elements are superimposed on the infinite, elastic, host rock mass where non-elastic behaviour is desired. Of course, if the yielding area is not known in advance, a reasonable guess must be made for the size of the plastic zone.

The mathematical theory required for introducing plasticity is fairly complex, and again, the reference is made to Banerjee and Butterfield(1981) and Volume 2.

In the solution process described in Section 2.6 the stresses are redistributed such that the safety factor is not less than unity in non-elastic volume elements. In this process load changes are applied in steps. When the stresses in a volume element exceed the strength criteria by a small amount an estimate of the increment of plastic deformation required to restore the stress state to the closest point on the failure envelope is calculated.

2.3 Multiple Materials

In MINE2D volume elements are used to represent different materials as well as plastic regions. By default volume elements have the properties of material 1. Alternate materials may be assigned to zones of volume elements in the first mining step.

In MINE3D a more efficient approach was developed for the three-dimensional formulation in MINE3D whereby different materials were treated as separate boundary element domains with interactions only through their boundaries (see Volumes 2 and 6).

2.4 Sequential Mining

Sequential mining has been introduced in the code to calculate the stresses, displacements, etc. at progressive stages of the mining operation. The input is prepared for a sequence of mining steps. At each new step, existing excavations are enlarged, new ones created, support pressures applied and/or backfill set in place.

Each new step is viewed as a modification of the former one. This eases the input process and speeds up intermediate calculations by providing at the new step "good" first guesses for unknowns involved in the iterative processes based on values obtained at the former step.

When all materials in the medium behave elastically, the solution at a particular mining step may be determined independently. For the case in which plasticity is involved, the mining history is taken into account by using the plastic stress solution from the previous mining step as the initial guess for the plastic initial stress field at the new mining step.

2.5 Fault Slip - Seam Mining Simulation

Fault slip simulation is accomplished by superimposing displacement discontinuities on the elastic host. A procedure similar to that developed by Crouch (1976) has been used here which enables fault slip and seam mining to be approximated.

It is assumed that the material behaves elastically until the stresses reach the peak strength defined by the strength criterion. Plastic deformations occur beyond this point.

The shear stress on the fault is not permitted to exceed the shear strength given by the Mohr-Coulomb criteria. The normal stress is not permitted to go below the tensile strength or to exceed the compressive strength. This latter limit is imposed to permit the simulation of pillar yielding in tabular mining problems.

Gouge or Seam Material

In order to simulate a seam or gouge material, the relationship between the surface stresses and displacement discontinuities are adjusted to force behaviour corresponding to a fictitious gouge or seam material present in the interface.

2.6 Solution Process

Solving problems with MINE2D is a multi-step process which can be described as follows:

1. The geometry is read into MINE, and the boundary element coefficient matrix is assembled.

2. The boundary conditions for the first mining step are set.

3. The first load step is begun by assuming that there is elastic behaviour only (i.e. no non-elastic response is permitted).

4. The coefficient matrix is solved using Gauss-Seidel iteration with relaxation. The matrix is considered to be solved when the stress error is reduced to a value below the stress tolerance.

5. The current stress state is checked at all points to test for failure. If failure is detected, an increment of permanent plastic deformation is applied.

6. A new load step is started, and control is returned to step 4 of this solution process.

7. The loop described here as steps 4 through 6 is repeated until the stress state at all points agrees with the failure criteria to within the specified stress tolerance.

8. A new mining step is started, and control is returned to step 4.

9. The loop described here as steps 4 through 8 is repeated until no additional mining steps are specified in the input file.

In order to control this solution process, several parameters must be set in the input file as described in 4.2.2 Control Parameters.

3.0 NUMERICAL MODELLING WITH MINE2D

3.1 Software Overview

MINE2D is written in FORTRAN77 and was originally developed on a DEC Microvax 3900 minicomputer. It has been converted for use on an 80386-type personal computer with a VGA screen and a 80387 or Weitek math coprocessor. This version was prepared using the SVS FORTRAN 77 compiler, the Phar Lap 386/DOS-Extender Software Development Kit, and Phar Lap 386/VMM (virtual memory manager).

MINE2D consists of two separate programs: MINE, the main analysis program, and OUTPUT, a graphical processor for checking the input model and plotting the calculated results.

In addition to the software supplied the user will require editor or word processor software to prepare the input data file, 386/DOS Extender software, 386/VMM software (large versions of MINE), and screen print software for plot hardcopy.

Various versions of the programs have been compiled for different problem sizes and for both the Intel 80387 or Wietek 1167 math coprocessors. Guidelines for software installation may be found by reading the README file on the distribution diskette.

3.2 Program Requirements and Limitations

In two common block files COMMON.BEM and COMMON.OUT that are part of the MINE2d source code, 5 parameters control the program dimensions as follows:

i) nd=2000, the total maximum number of control points;

ii) ndcs=4000, the total maximum number of degrees of freedom;

iii) ndg=10, the total maximum number of field point grids;

iv) ndm=10, the total maximum number of materials; and,

v) ndp=3000, the total maximum number of grid points.

To change the program dimensions these files must be changed and the source code recompiled.

The OUTPUT program on the software distribution diskette has been compiled with these parameters set as shown above and requires a computer with 2 megabytes of RAM.

The MINE program requires a considerable amount of memory in order to form the coefficient arrays needed for the boundary element solution. A minimum of 4 megabytes of RAM is required for MINE and 8 to 16 megabytes is recommended. The 16 megabyte version of MINE used for the example problem in Section 7.0 is recommended for learning to use the program. If insufficient RAM memory is available for the required problem size MINE may be installed to run with the virtual memory manager so that the additional memory requirement is met from the hard disk drive. The memory requirement and problem size for several versions of MINE that have been compiled are given in Table 3-1.

Version	Memory Req'd	Nd Max. # Control Points	Ndcs Max. # Degrees Freedom	Approximate Number of Zone Subdivisions That Can be Modelled (based on Type 3 Zones)
M2000	134 Mb	2000	4000	1100
M1200	46 Mb	1200	2400	680
M675	16 Mb	675	1350	380
M450	8 Mb	450	900	190
M300	4 Mb	300	600	125

Table 3-1 Memory Requirements and Problem Size for MINE

The detailed calculation of the number of control points and the number of degrees of freedom required for a MINE2D problem is discussed in Section 4.2.3 Zone Geometry Definition.

Plane Strain Limitation

The formulation for MINE2D assumes a two-dimensional plane strain state. Plane strain is associated with long excavations of constant cross-section through the model and acted on by loads in the plane of the cross-section. While this restricts the mining problems that can be accurately modelled, MINE2D may be useful to compare alternate plans where non-elastic behaviour is important.

3.3 Modelling Sequence

Problem analysis with MINE2D is divided into the following stages:

1. Prepare a Plan of the Model Geometry - This should indicate the zones that will be used to define regions of non-elastic behaviour, excavations, backfill, and faults. All conditions and parameters, for each mining step which is to be analyzed, must be defined before the data file is created.

2. Generate Input Data File - A text file editor such as SPF/PC or a word processor with the ability to import and export ASCII files is used to prepare a file "model.INP" in the prescribed format. The input parameters are described in Section 4.2 .

3. View Input Data with OUTPUT - The data file should be checked before the analysis is performed to ensure the model geometry displayed is correct at each mining step.

4. Perform Analysis with MINE - The data file is read into MINE, the coefficient arrays are assembled, and the iterative solution process is repeated for each mining step. The iteration count and solution error are printed to the screen so that the user may monitor the solution process. Convergence issues are discussed in Section 5.2

5. Plot the Results with OUTPUT - The calculated data from MINE is read into OUTPUT for the specified mining steps. Plots of stress, displacement, strain, safety factor, failure(yield) zones, and stored strain energy density can be viewed as described in Section 6.

6. Evaluate Printed and Graphics Results - If the factor of safety plot indicates failure beyond the user-selected plasticity region more plastic elements should be added in those areas and the model re-run. The calculated incremental energy release for each mining step is printed in the file "model.OUT".

7. Return to stage 2 to evaluate different parameter variations and plans. Restart from any of the previously calculated mining steps to continue the mining sequence or study alternative sequences.

3.4 Energy Calculation

At the completion of each mining step, the incremental energy release in the mining step is calculated in MINE. This data together with a summary of the mining sequence is printed to the file "model.OUT". The energy components presented are defined as follows:

We - energy associated with the change in stress acting on the surface of excavations (elastic unloading),
Wc - elastic strain energy stored and released,
Wb - energy used in the deformation of backfill,
Ws - energy used in the deformation of supports,
Wp - energy expended in the plastic deformation of material,
Wv - work done in viscous deformation,
Um - energy removed with the excavation of material of volume Vm,

These contributions can be summed as followed to determine the excess energy for two different assumptions:

i) excess energy assuming controlled elastic unloading of surfaces

$$Wk_elastic\ unloading = We - (Wc + Wp + Wb + Ws)$$

ii) excess energy assuming instantaneous unloading of surfaces

$$Wk_instant\ unloading = - (Wc + Wp + Wb + Ws)$$

Note that the component of energy Wv, associated with the visco-plastic deformation has not been included in this summation since it represents an artificial energy sink imposed only to control the deformation rates.

A detailed discussion of the exact meaning of these energy terms is presented in Volume 2 of this report.

Since MINE2D is a two-dimensional model, work done and energy are expressed in terms of energy per unit of depth. Units and calculation of the actual energy term are discussed in the next section (3.5 Sign Convention and Units).

3.5 Sign Convention and Units

The following sign conventions are used in MINE2D and must be kept in mind when entering input.

MINE2D assumes that compressive stresses and strains are positive, and tensile stress and strains are negative. Displacements are positive in the direction of the axes.

All strength parameters should be given with regards to correct sign, i.e. compressive strength and cohesion positive, tensile strength negative.

Far field stresses should be specified as positive values.

Recommended units for both SI and British systems are in Table 3-2.

ITEM	METRIC UNIT	BRITISH UNIT
Length	m	ft
Stress	MPa	psi
Modulus	MPa	psi
Displacement	m	ft
Strain	microstrain	microstrain
Work Done	MJ/m=MN	psi.ft**2
Energy	MJ/m=MN	psi.ft**2

Table 3-2 System of Units

Note that since MINE2D is a two-dimensional model, work done and energy are expressed in terms of energy per unit of depth. Multiply by the depth (normal to the section used for analysis) of the problem to obtain the actual energy term:

i) In metric units, multiply by the depth in metres to obtain Mega-Joules;

ii) In British units, the use if psi instead of psf units for stress requires that the results must first be multiplied by 144 to convert to units of Foot-Pounds/ft. Then multiply by the depth in feet to obtain Foot-Pounds.

3.6 Hardcopy Plots

No hardcopy of result plots is provided directly by the OUTPUT program. Hardcopy plots are obtained by the use of third party Print Screen programs such as PIZAZZ PLUS or INSET. Install these routines as instructed and press <PrtSc> to print the screen plot directly to the printer or to a file for latter use. The color

result plots for the example problem in Section 7 were printed on a HP Paintjet printer using PIZAZZ PLUS.

4.0 INPUT PREPARATION

4.1 Planning the Model

In MINE2D, one starts with an infinite elastic, host rock mass, and any features such as excavations, non-elastic zones, alternate materials, or faults must be superimposed. There are four zone types available to describe such features. These include fictitious force boundary elements, displacement discontinuities, volume elements, and excavation elements.

These zone types can be combined to build up complex mining geometries which are intersected by faults and joints. Note that in all cases, any contact between zones must occur along the edges of the zones such that the adjacent zones share common corner points. Also, along such adjacent edges, the zones must have an equal number of subdivisions.

An plan prepared for the example problem is shown in Figure 7-1.

4.2 Preparing the Input Data File

A text file editor such as SPF/PC or a word processor with the ability to import and export ASCII files is required to prepare a data input file "model.INP" in the prescribed format. This is normally done by editing a copy of an existing input file such as "SAMPLE.INP" which is distributed with the program. Copy the sample input file to "model.INP" where "model" is the desired model name of up to 8 characters which will also be assigned to result files. Edit the "model.INP" to describe the desired model.

The input data file is grouped into 8 parameter sets, as shown in Figure 4-1. Each parameter set must be complete, in the prescribed order, and separated from the other sets by **0/** . The first 7 sets define the model parameters and geometry common to all mining steps. The 8th set defines the change at a given mining step. This set is repeated for each mining step. Guidelines for completing these parameter sets are presented in Sections 4.2.1 to 4.2.9.

Comments lines may be inserted in the file by placing a * in the first column of the line. **Blank lines are not permitted.**

```
* JOB DESCRIPTION SECTION ( used in output plot headings )
  SAMPLE MINE2D MODEL
* CONTROL PARAMETERS SECTION
* |----------|----------|--------------------|----------|----------|
* |Max. # of |Max. # of |Relaxation Parameter |  Stress  | Symmetry |
* |Load Steps|Iterations|   Maximum | Minimum|Tolerance | Planes   |
* |/Mng. Step|/Load Step|           |        |(psi/MPa) |          |
* |----------|----------|-----------|--------|----------|----------|
        1000       1000       1.200    0.200    5.0000         0
* ZONE GEOMETRY DEFINITION LIST
* |----|-----------------|--------|----|---------|--------------------|
* |Zone| Coordinate Number | # of Div|    |Thickness|                    |
* | Num| Pt.| Pt.| Pt.| Pt.|Between |Type|( type 2)|    Description     |
* |    |  1 |  2 |  3 |  4 |1-2| 2-3|    |         |    (optional)      |
* |----|----|----|----|----|---|----|----|---------|--------------------|
     1    2    3   14   15    7    7    3      0.00 'description'
    0/
* COORDINATE LIST
* |---------|-----------|-----------|
* |  Coord  |     X     |     Y     |        FOR ALL POINTS REQUIRED
* |  Number |   Coord   |   Coord   |        IN ZONE GEOMETRY AND
* |---------|-----------|-----------|        GRID DEFINITION
          2     1100.00     1150.00
          0/
* STRESS STATES
* |---------|-----------------------------------------|---------|
* |Material |          Principal Stresses             |         |
* | Number  |Sigma:1 = a * y + b|Sigma:2 = a * y + b  |  Theta  |
* |         |----a----|----b---|----a----|----b---- |         |
* |---------|--------|--------|---------|----------|---------|
          1      0.00   5200.00      0.00   3350.00     90.00
          0/
* MATERIAL PROPERTIES LIST
* |---------|------- Peak Strength Properties / Viscous Parameters -------|
* |Num |Type|     E     | Poisson's |    To     |    Co     |   So,m    |  phi,s    |
* |    |    |   dummy   |   dummy   |   dummy   |   dummy   |   dummy   |   dummy   |
* |    |    |    G1     |    G2     |   dummy   |   dummy   |   dummy   |   dummy   |
* |----|----|-----------|-----------|-----------|-----------|-----------|-----------|
    1   +1 1.0000E+07        0.30    -300.00    9000.00        0.00       45.00
           0.0               0.00       0.00       0.00        0.00        0.00
           1.0000E+06        0.00       0.00       0.00        0.00        0.00
   0/
* GRID DEFINITION LIST
* |-------|--------------------------|-------------|------------|
* | Grid  |     Coordinate Number    | # of Div    |            |
* |Number | Pt.  | Pt.  | Pt.  | Pt. |Between Pt.|  Material  |
* |       | One  | Two  |Three | Four| 1-2 | 2-3 |  Number    |
* |-------|------|------|------|-----|-----|-----|------------|
        1    101    102    103    104     36     48       1
       0/
* MINING STEPS
* |---------|---------|
* |  Zone   |         |
* | (block) |  Code   |
* |---------|---------|
* Mining Step 1 place material 2 in zone 2 and excavate zone 1
          2        +2
          1         0
          0/
```

Figure 4-1 MINE2D Input Data File Format

4.2.1 Job Description

This heading is printed on all graphics plots.

4.2.2 Control Parameters

A set of 6 control parameters must be defined.

Max. # of Load Steps/Mining Step limits the maximum number of times the load step loop will be executed for any given mining step. Proper execution requires that the stress state at all points agrees with the failure criteria to within the specified stress tolerance before this limit is reached. If this limit is reached, it is assumed that the problem is unsolvable, and MINE2D is stopped.

It is recommended that a large number (1000 or more) be specified for this parameter.

Max. # of Iterations/Load Step limits the maximum number of Gauss-Seidel iterations for each load step. Proper execution requires that the stress error falls below the stress tolerance before this limit is reached. If this limit is reached, it is assumed that the problem is unsolvable, and MINE2D is stopped.

It is recommended that a large number (1000 or more) be specified for this parameter.

Maximum and **Minimum Relaxation Parameter** limit the range over which the relaxation parameter is allowed to vary. The relaxation parameter is used in Gauss-Seidel iterations to speed convergence. During the solution process, the relaxation parameter is adjusted depending on the how the solution is proceeding.

If the stress error is steadily reducing towards zero, the relaxation parameter is increased closer to the maximum.

If the stress error is erratic or increasing, the relaxation parameter is reduced towards the minimum.

It is recommended that the minimum relaxation parameter be set to 0.2, and the maximum to 1.2 .

Stress Tolerance is the maximum allowable stress error. Both Gauss-Seidel iterations (step 4 in Section 2.6) and load stepping (steps 4 through 6 in Section 2.6) continue until the stress error falls below the stress tolerance.

It is recommended that the stress tolerance be set to a value of approximately 0.1% of the far field stress magnitude.

Symmetry Planes is the number of planes of symmetry which will be used by MINE2D as follows:

Use **0** for no symmetry.

Use **1** to cause the entire geometry to be symmetric about the y axis such that the x components are duplicated with -x.

Use **2** to cause the entire geometry to be symmetric about both the x and y axes such that both the x and y components are duplicated with -x and -y.

If symmetry is used, the far field stress state must be aligned parallel to the x and y axes.

4.2.3 Zone Geometry Definition

All zones that will be required to describe features such as excavations, non-elastic zones or faults in the series of mining steps must be defined.

Zone Number assigns a unique identification number to each zone.

Coordinate Numbers are the coordinate numbers of points from the coordinate list that define the end points or corners of zones. The requirements for different zone types are described below in the description of zone types.

Number of Divisions

The number of divisions must also be specified. This controls the number of elements that the zones will be subdivided into for analysis purposes. Normally five subdivisions per side of an excavation is the minimum necessary to achieve reasonable accuracy. More subdivisions may be required on long sides, or where detailed results are required.

A large enough number should be specified so that the subdivided element length is similar to other element lengths in the vicinity. If convergence problems are occurring, often increasing the number of subdivisions will help.

# of Divisions 1-2 is the number of elements the zone is to be subdivided into between Point 1 and Point 2.

# of Divisions 2-3 is the number of elements the zone is to be subdivided into between Point 2 and Point 3.

If the number of subdivisions on side 1-2 is n, type 1 and type 2 zones are subdivided into n equally spaced elements with a total of

n+1 control points. This contributes 2.(n+1) degrees of freedom to the total.

If the number of subdivisions on side 1-2 is n, and on side 2-3 is m, type 3 and type 4 zones are subdivided into an n by m grid of equally spaced elements with a total of (n+1).(m+1) control points. This contributes 3.(n+1).(m+1) degrees of freedom to the total.

Surface elements are also generated around the periphery of type 3 and type 4 zones, contributing an additional 2.(n+1+m+1) control points, and 4.(n+1+m+1) degrees of freedom to the total.

Type 1 Zones

Type 1 zones define a straight line of standard fictitious force boundary elements. This zone type is used to describe the surfaces of excavations. These zones are specified by listing the coordinate numbers of the two end points (Pt.1,Pt.2) of each zone. Zones must be ordered such that excavations are traversed around their boundary surface in a counterclockwise sense.

Type 1 zones are permitted to connect to other type 1, or type 2 zones provided all connections are made at common end points. Intersection or connection to other zones in any other way is not permitted. Connections to type 3 or type 4 zones is not recommended.

Type 4 zones can also be used to make excavations.

Type 2 Zones

Type 2 zones define a straight line of displacement discontinuity boundary elements. These zones are used to describe faults and joints, or tabular mining. They are specified by listing the coordinates of the two end points (Pt.1,Pt.2) of each zone. It is necessary to supply a thickness for these zones. This parameter is used in the determination of the elastic response of any gouge material present within these zones.

Type 2 zones are permitted to connect to other type 1 or type 2 zones provided all connections are made at common end points. They are not permitted to intersect other zones in any other way.

Type 2 zones may also connect to, or run along the edge of other type 3 or type 4 zones provided all zones share common end or corner points. Also, when run along the edge of type 3 or type 4 zones, the same number of subdivisions must be used for both zones along the common edge. Type 2 zones are not permitted to run across or intersect type 3 or type 4 zones in any other way.

Type 3 Zones

Type 3 zones define a quadrilateral or triangular zone of volume elements. Within type 3 zones, the material can behave non-elastically, and with an elastic stiffness different than that of the host rock mass. These zones are used to describe areas of material type other than the elastic host rock mass, such as stiff dykes, or backfilled stopes. These zones can also be used to create excavations.

Type 3 zones are specified by listing the coordinates of the three or four corners of each zone. These zones may connect to other type 3 or type 4 zones provided the adjacent zones share common corner points, and the same number of subdivisions are used along the common edge. These zones are not permitted to overlap or intersect other zones in any other way.

Type 4 Zones

Type 4 zones define regions of volume elements which appear identical to type 3 zones. However, unlike type 3 zones, the material within these zones behaves exactly the same as the elastic host rock mass. These zones are only used to create excavations.

These zones are specified by listing the coordinates of the three or four corners of each zone. These zones may connect to other type 3 or type 4 zones provided the adjacent zones share common corner points, and the same number of subdivisions are used along the common edge. These zones are not permitted to overlap or intersect other zones in any other way.

Thickness is required for zone type 2. Specify **0** for other types.

Description is an optional data input field which may be left blank. Descriptions input in single quotes will be shown in the listing of mining steps in filename.OUT when alternate material is placed in the zone or it is excavated.

4.2.4 Coordinate List

This list provides the x and y coordinate locations of each coordinate point. Coordinate points are used to specify the geometry of the zones and data grids.

Coordinate Number is the unique identification number to be assigned to the point. The coordinate number list does not have to be sequential or continuous. Do not assign more than one coordinate number to the same point.

X-Coordinate and **Y-Coordinate** are the cartesian coordinates of the point.

4.2.5 Stress States

This list provides a set of stress states. These are used to specify the far field stress state for the host rock mass, as well as to insert materials into excavations under initial stresses (self-weight), or to assign stresses to excavation surfaces. Stress states are specified by giving the principal stress components and their orientation. These stresses may vary with depth. These are assigned to specific zones in the mining step section.

State Number is the unique identification number assigned to each stress state defined. Stress state number 1 is reserved for specification of the far field stress state of the host rock mass.

Stress state numbers 2, 3, and so on, are associated with material numbers 2, 3, and so on (respectively).

Principal Stress magnitudes for **Sigma:1** and **Sigma:2** are specified by the components **a** and **b**.

$$\text{principal stress magnitude} = (a * y) + b$$

If the stresses are not to vary with depth (y) set a to 0 and set b equal to the constant stress desired.

Theta is the angle (degrees) measured counterclockwise from the positive X-axis to the orientation of Sigma:1.

A total maximum of 10 stress states can be used.

4.2.6 Material Properties - Failure Criteria

This list provides a set of material properties, used to specify the elastic response of the host rock mass, as well as any materials inserted into type 2 or type 3 zones. These are assigned to specific zones in the mining step section.

Three parameter lines are requires for each material. The first provides the material number, material type and peak strength properties. The second line was intended for residual strength properties which are not considered in MINE2D and must contain 6 dummy values (ie. 0.0). The third line contains the viscous parameters G1 and G2 plus 4 dummy values.

Material Number is the unique identification number assigned to each set of material properties. Material number 1 is reserved for the host rock mass. Materials 2, 3, and so on are used to specify other sets of material properties, which may be inserted into type 2 or type 3 zones.

Material Type codes specify the material behaviour that will be allowed and the failure criteria (Mohr-Coulomb or Hoek-Brown) that will be used as follows:

Code	Material Behaviour	Failure Criteria
0	Elastic	(no failure or Safety Factor)
1	Elasto-Visco-Plastic	Mohr/Coulomb
2	Elasto-Visco-Plastic	Hoek/Brown
-1	Visco-Plastic	Mohr/Coulomb
-2	Visco-Plastic	Hoek/Brown

By definition, the host rock mass (material 1) is only permitted to behave elastically. However, the material type parameter should be set to 1 or 2 for this material to indicate which failure criteria (Mohr-Coulomb or Hoek-Brown) to use during the contouring of safety factors on field point grids.

Material in Type 2 Zones

For type 2 zones, the material type parameter may be set either to 0, -1, or 1. Material type 0 indicates that the material behaves in an elastic manner only, i.e. no fault slip, crushing or crack opening. Material type -1 indicates that the material behaves in a rigid visco-plastic manner only, i.e. fault slip, etc., but no elastic response. Material type 1 indicates that the material behaves in an elasto-visco-plastic manner, i.e. elastic response up to failure, followed by fault slip etc.

If either 0 or 1 is specified for the material type, elastic properties must be given.

If either -1 or 1 is specified, Mohr-Coulomb failure criteria parameters must be given. This criteria has three bounds:

i) normal stress is not permitted to go below the tensile strength To;

ii) shear stress is not permitted to exceed the shear strength given by the Mohr-Coulomb criteria

$$So + (Sigma:n * \tan(phi))$$

where So represents the cohesion intercept, and phi represents the friction angle of the fault; and,

iii) the normal stress is not permitted to exceed the compressive strength Co. This latter limit is imposed to permit simulation of pillar yielding in tabular mining problems.

For simulation of joints or faults, normally both To and So would be set to zero, and Co would be set to a very large number to disable any compressive failure. Phi would probably be set in the range from 20 to 30 degrees. Care should be taken to ensure that To is not lower than the intercept of the Mohr-Coulomb criteria with the normal stress axis, i.e.

$$To > -So/\tan(phi)$$

For tabular mining, Co would be set equal to any reasonable field scale pillar strength.

Material in Type 3 Zones

For type 3 zones, the material type parameter may be set either to 0, -1, 1, -2 or 2. Material type 0 indicates that the material behaves in an elastic manner only, i.e. no failure. Material type -1 or -2 indicates that the material behaves in a rigid visco-plastic manner only, i.e. failure but no elastic response. Material type 1 or 2 indicates that the material behaves in an elasto-visco-plastic manner, i.e. elastic response up to failure, followed by yielding.

If either 0, 1 or 2 is specified for the material type, elastic properties must be given.

If either -1 or 1 are specified, Mohr-Coulomb failure criteria parameters must be given. This criteria has two bounds:

i) the minor principal stress is not permitted to go below the tensile strength To; and,

ii) the major principal stress is not permitted to exceed the strength given by the Mohr-Coulomb criteria;

```
Co + (S3 * [tan(45+phi/2)]**2)
```

where Co represents the field scale compressive strength, and phi represents the friction angle. These parameters should be set to values those representative of the strength on a rock mass scale. Care should be taken to ensure that To is not lower than the intercept of the Mohr-Coulomb criteria with the minor principal stress axis, i.e.

```
To > -Co/[tan(45+phi/2)]**2
```

If either -2 or 2 are specified, Hoek-Brown failure criteria parameters must be given. This failure criteria has two bounds:

i) the minor principal stress is not permitted to go below the tensile strength To; and,

ii) the major principal stress is not permitted to exceed the strength given by the Hoek-Brown criteria;

```
S3 + [(m * Co * S3) + (s * Co**2) ]**(1/2)
```

where Co represents the laboratory scale compressive strength, and m and s are Hoek-Brown parameters. The parameters used here should be selected according to the guidelines recommended by Hoek and Brown (1988), or determined from back analysis. Care should be taken to ensure that To is not lower than the intercept of the Hoek-Brown criteria with the minor principal stress axis, i.e.

```
To > (Co/2) * ( m - [ m**2 + (4 * s) ]**(1/2) )
```

Viscous Parameters

If the material type parameter is non-zero, viscous response can be controlled with the Visco-Plasticity parameters G1 and G2. These parameters limit the amount of non-elastic deformation which is permitted to occur during any load step.

If a small or zero value is specified for both parameters, there will be limited viscous resistance, and MINE2D will attempt to apply all non-elastic deformation in one load step. This will normally result in unstable behaviour, and a solution to the problem will not be possible.

If too large a value is specified, there will be too much viscous resistance, and non-elastic deformation will occur very slowly. A very large number of load steps will be required to reach a solution to the problem.

An exact value cannot be recommended, however a good first guess is to set G1 equal to a value of the same order of magnitude as Young's modulus, and set G2 to zero. If the solution progresses very slowly, and too many load steps are used, try a smaller value for G1. Alternatively, if the solution process is very erratic or unstable, try a larger value. G2 could be experimented with in the same way.

4.2.7 Grid Definition

Field point grids are specified exactly the same way as type 3 or type 4 zones: the coordinate points of the four corners of the grid are listed. Grids are optional, since results are always calculated inside type 3 and 4 zones when they contain material. Grids are permitted to overlap zones. During the contouring process, any results within type 3 or type 4 zones are drawn over top of the grid results. Where grid points are outside such zones, results will be calculated using the strength criteria of the material indicated.

The number of divisions must also be specified. This controls the number of field points that the grid will be subdivided into for analysis purposes. Normally enough subdivisions should be specified so that the spacing of the field points is similar to element lengths in the vicinity.

If the number of subdivisions on side 1-2 is n, and on side 2-3 is m, the grid is subdivided into a total of (n+1).(m+1) grid points. A total maximum of 3000 grid points can be used.

One can also specify which material property number will be used for safety factor calculations during contouring of the results.

A total maximum of 10 grids can be used.

4.2.8 Mining Steps

This part of the input file controls the mining sequence by permitting users to assign a variety of boundary conditions to the zones. A normal mining sequence would consist of an initialization step, where material properties where assigned to zones such as stiff dykes of gouge in faults. In this step, some zones could be excavated as well. Subsequent steps could include placement of backfill into previously excavated stopes, and additional excavation.

There is no limit to the number of mining steps which can be used.

Excavations With or Without Surface Stresses

Excavations are specified by listing the zone number, followed by either a zero or negative number for the code. A zero indicates that the surface of the excavation is stress free. Alternatively, if a negative number is specified for the code, stresses are applied to the surfaces of the excavation. The magnitude of these stresses is determined from values specified in the stress state list (see section 4.2.5). For example, if a -3 is specified for the code, stress state number 3 in the stress state list is used.

Specification of surface stresses is useful for prestressing excavation surfaces to simulate the placement of backfill at a known stress (e.g. equivalent to the weight of the backfill). This feature can also be used to simulate pressurized openings.

All zone types can be used in this way. For zone type 1, the user must ensure that a loop of elements describing a complete excavation is defined. A zone type 2, excavation would only be used to simulate tabular mining. For zone types 3 and 4, complex excavation shapes can be constructed by excavating adjacent zones simultaneously.

Inserting Materials Into Type 2 or Type 3 Zones

To insert a material into either a type 2 or type 3 zone, list the zone number followed by the material number (positive value). The material is placed in the specified zone with the initial stress state specified in the stress state list.

For faults, inclusions such as stiff dykes, or tabular mining, one would normally specify that material with an initial stress state exactly equal to the far field stress state of the host material. This would be done for material 2 for example, by specifying a positive two (2) for the code. In this case, stress state number 2 should be specified with the same stresses as stress state one (the host far field stress state).

Other materials such as backfill are normally placed at stress states equivalent to there own weight only. This would be done for material 3 for example, by specifying a positive three (3) for the code. In this case, stress state number 3 should be specified with stresses representative of the weight of the backfill.

Materials cannot be placed into type 1 or type 4 zones.

4.2.9 Restarting Result Calculation

An analysis can be restarted at any given mining step which was either completed, or incomplete because a final converged solution was never obtained. This is done by specifying a negative value in place of the zone number. The value refers to the number of the mining step.

This feature is useful for testing alternate mining sequences, or restarting a problem which failed to converge.

To instruct the MINE program to restart calculations after a previously solved mining step, n, edit the input file. Specify -n as the only operation in the first mining step and define the next mining step(s) to be performed as shown in Figure 4-2. In order to restart the analysis from mining step "n", the result files name.OUT and name.00n from the previous analysis are required. If alternate mining sequences are to be tested copy these previous result files to the same name as the alternate input file.

```
* MINING STEPS
*|---------|---------|
*|  Zone   |         |
*| (block) |  Code   |
*|---------|---------|
*  Restart from results of Mining Step  2
            -2
             0/
* Mining Step   3   excavate panel 4
             4          0
             0/
```

Figure 4-2 Example Mining Steps to Restart MINE2D

5.0 MINE2D ANALYSIS PROGRAM (MINE)

5.1 Running the Analysis

The MINE program performs the boundary element calculations using the data contained in the previously prepared input data file "model.INP". Start the analysis by entering the command MINE and the model input file name, including the ".inp" file type, as shown in Figure 5-1. If the input data is successfully read into MINE and checked for completeness the user may type "Y" to proceed with the analysis or "N" to exit the program. Analysis progress is reported by messages on the screen as illustrated in Figure 5-2.

An "End of File Encountered While Reading Data" message indicates that the file has not been successfully read because of blank lines or the incorrect number or type of values on a line. Edit the input file to conform to the format prescribed in Section 4.

As described in Section 3.2 various problem size versions of MINE have been prepared to make use of the available computer memory and/or disk space. The problem size is determined during the assembly process. If the maximum number if control points or the maximum number of degrees of freedom are exceeded an error message will report the condition and the program will stop. Decrease to number of elements in the model and try again.

Monitoring the iteration process is discussed in Section 5.2

```
C:\MINE2D> MINE <Enter>
Start Program MINE
¦-----File Name (*.inp)-------------------¦
 model.inp <enter>

File successfully opened
File successfully read
-------------------------------------------------------------------------------
Restart at mining step     2
Restart at loading step   38
-------------------------------------------------------------------------------
Job Title           SAMPLE MINE2D MODEL
Mining Step            2
    Analysis
-------------------------------------------------------------------------------
Job Title           SAMPLE MINE2D MODEL
Mining Step            3
    Block     2, insert material number      4,
    Block     3, excavate to stress state    0,
    Analysis
-------------------------------------------------------------------------------
Do you want to analyze this problem (Y/N) ?
Y
```

Figure 5-1 Starting MINE Analysis Program

```
Input Complete
Preparation Complete
Assemble element     1 of  561 ...
Matrix Assembly Complete
-------------------------------------------------------------------------------
Restart at mining step     2
Restart at loading step   38
-------------------------------------------------------------------------------
Program MINE 1.01
Job Title           SAMPLE MINE2D MODEL
Analysis Date      27-SEP-89
Analysis Time      11:00:52
Mining Step            2
    Analysis
-------------------------------------------------------------------------------
Iteration #     1, Rpp =1.10, Error =         0.78 Ferror         0.78  211
Time Step   39 is complete

Calculate field point      1 of 1287 ...
Convergence achieved after   39 time steps Error =  0.78 Ferror =      0.78
Mining Step    2 is complete
-------------------------------------------------------------------------------
Program MINE 1.01
Job Title           SAMPLE MINE2D MODEL
Analysis Date      27-SEP-89
Analysis Time      11:00:52
Mining Step            3
    Block      2, insert material number       4,
    Block      3, excavate to stress state     0,
    Analysis
-------------------------------------------------------------------------------
Iteration #   30, Rpp =1.20, Error =        24.83 Ferror     12873.65  401
Time Step    0 is complete
Iteration #    2, Rpp =1.10, Error =     10244.53 Ferror     12047.34  254
Time Step    1 is complete
   ...
Time Step   27 is complete
Iteration #    1, Rpp =1.10, Error =        30.27 Ferror        30.27  246
Iteration #    2, Rpp =1.10, Error =        23.99 Ferror        21.85  406
Time Step   28 is complete

Calculate field point      1 of 1287
 etc.
Calculate field point  1287 of 1287
Convergence achieved after   28 time steps Error =  23.99 Ferror =   21.85
Mining Step    3 is complete
-------------------------------------------------------------------------------
   ... (repeated for each mining step)
End Program MINE
Your energy summary is stored in the file named
SAMPLE.out
To generate contour plots of the results
Enter OUTPUT
Normal Termination
```

Figure 5-2 MINE Analysis Monitoring Display

5.2 Convergence Issues

In the Section 4.2, numerous references have been made to the parameters which control the solution to problems. These include:

i) Maximum # Load Steps per Mining Step;
ii) Maximum # of Iterations per Load Step;
iii) Relaxation Parameters;
iv) Stress Tolerance; and,
v) Viscous Parameters.

During the running of the analysis program MINE, information regarding the solution process is printed to the computer screen as shown in Figure 5-2. This includes a log of program activity, but more important, parameters that monitor the progress towards solution convergence are also presented. These include:

i) Iteration # - the current iteration number in the matrix solution;

ii) Rpp - the current relaxation parameter in the matrix solution;

iii) Error - the current stress error, indicated how well the coefficient matrix is satisfied (note that this parameter may take on negative or positive values);

iv) Ferror - the current stress error, indicating how well the failure criteria at all points are satisfied (note that this parameter only takes on positive values, and only when the failure criteria is exceeded at some point); and,

v) Time Step - the current load step number.

For a given mining step, a solution is considered to be converged when the magnitude of both the Error, and the Ferror are below the stress tolerance.

5.2.1 Coefficient Matrix Solution

The first three parameters (i, ii and iii) indicate how the Gauss-Seidel iterative solution of the coefficient matrix is proceeding. A large number of iterations (greater than a few hundred) indicates that the problem is poorly conditioned and difficult to solve. This normally only occurs due to incorrectly specified parameters, or due to a poorly constructed geometry. Although, this condition may also occur for problems where faults intersect excavations at low angles. Using a larger number of subdivisions on these faults in the vicinity of the excavation can improve this situation.

If the solution is proceeding asymptotically towards zero Error, and a large number of iterations are required for solution, one

should check to ensure that the maximum relaxation parameter is set high enough (i.e. it should be in the range from 0.8 to 1.2), and also that the minimum relaxation parameter is not set too low (i.e. it should be in the range form 0.2 to 0.5).

Another parameter to check is the stress tolerance. The smaller this parameter is set, the larger the number of iterations which will be required for solution. Although this parameter controls the accuracy of the solution, excessively small values are not realistic, and will lead to unnecessarily large numbers of iterations. The stress tolerance should be set at about 0.1% of the far field stress magnitude.

If the solution is erratic in that the Error is oscillating or jumping around in magnitude, there are two possible causes:

i) if Ferror is zero, then one should reduce the maximum relaxation parameter to a value nearer 0.5; or,

ii) if Ferror is non-zero, this condition is probably occurring because the viscous modulus G1 is set too low. In this case, MINE2D would be attempting to apply too much plastic deformation in one load step. This often leads to unstable conditions. Try increasing G1 by a factor of 10.

5.2.2 Load Step Solution

The last two parameters in the list above (Ferror and Time Step) indicate how the non-elastic solution is proceeding. If the solution is erratic in that the Ferror is oscillating or jumping around in magnitude, the viscous modulus G1 is probably set too low. In this case, MINE2D would be attempting to apply too much plastic deformation in one load step. This often leads to unstable conditions. Try increasing G1 by a factor of 10.

If the solution is proceeding asymptotically towards zero Ferror, and a large number of time steps (greater than a few hundred) are used, the value of the viscous parameter G1 is probably set too large. Try reducing the magnitude of G1 by one half or more.

Finally, if the solution is proceeding asymptotically towards some non-zero Ferror, and a large number of time steps (greater than a few hundred) are used, the problem is probably unsolvable. This kind of condition occurs in problems where a wedge is falling out of the hanging wall, or an unstable failure process has occurred. This kind of problem can only be solved by excavating the unstable zones, or applying a stabilizing internal pressure as might be supplied by backfill. Note however, that recognition of unstable failure conditions may be the objective of the analysis.

5.3 Result Files

The MINE program produces result files having the same filename as the input data file "model.INP". The files produced are as follows:

"model.OUT" contains a summary of the mining sequence and the incremental energy release data calculated by MINE for each mining step. The file contents are illustrated in Figure 5-3. The energy components are explained in Section 3.4. The file is written in ASCII format and can be examined with a file editor. This data is not presented by the graphic processor OUTPUT.

"model.000" is a scratch file that can be deleted (binary format).

"model.00n" is a result file written for each mining step, **n**. These files (model.001, model.002, etc.) contain the calculated data such as stresses and displacements to be read into the OUTPUT program. These files are also required by MINE when the analysis is to be restarted from the results of a previously solved mining step. These files are written in binary format to conserve file space and speed file access.

"model.TXT" contains calculated results at element nodes and data grid points for each mining step. For each node, the X, Y, Sxx, Syy, Sxy, shear displacement and normal displacement values are reported. For each data grid point, the X, Y, Sxx, Syy, Sxy, X displacement, and Y displacement values are reported. The file is written in ASCII format and usually requires considerable file space. The file is not used by OUTPUT to produce result plots and may be deleted if such detailed results are not required.

Section 6 describes the use of the graphical processor OUTPUT to generate plots of the calculated data.

```
-------------------------------------------------------------------------------
Restart at mining step     2, loading step    38
-------------------------------------------------------------------------------
Program MINE 1.01
Job Title           SAMPLE MINE2D MODEL
Analysis Date      27-SEP-89
Analysis Time      11:00:52
Mining Step            2
-------------------------------------------------------------------------------
 Energy Change up to loading step                              39
    Work Done (elastic unloading)   We          0.00000E+00
    Stored Strain Energy Change     Wc         -5.38206E-02
    Work Done on Backfill           Wb          0.00000E+00
    Work Done on Support Loads      Ws          0.00000E+00
    Plastic Work Done               Wp          3.47564E-02
    Viscous Work Done               Wv          4.45541E-02
    Excavated Volume                Vm          2.00000E+03
    Strain Energy in Vm             Um          8.17880E+03
    Wk_elastic unload We-(Wc+Wb+Ws+Wp)          1.90642E-02
    Wk_instant unload   -(Wc+Wb+Ws+Wp)          1.90642E-02
Convergence achieved after   39 time steps Error =   0.78 Ferror =  0.78
Mining Step    2 is complete
-------------------------------------------------------------------------------
Program MINE 1.01
Job Title           SAMPLE MINE2D MODEL
Analysis Date      27-SEP-89
Analysis Time      11:00:52
Mining Step            3
    Block     2, insert material number      4
    Block     3, excavate to stress state    0,
-------------------------------------------------------------------------------
 Energy Change for ELASTIC stress redistribution
    Work Done (elastic unloading)   We         -9.13541E+03
    Stored Strain Energy Change     Wc         -9.71733E+03
    Work Done on Backfill           Wb          9.45216E+01
    Work Done on Support Loads      Ws          0.00000E+00
    Plastic Work Done               Wp          0.00000E+00
    Viscous Work Done               Wv          0.00000E+00
    Excavated Volume                Vm          1.00000E+03
    Strain Energy in Vm             Um          7.58849E+03
    Wk_elastic unload We-(Wc+Wb+Ws+Wp)          4.87396E+02
    Wk_instant unload   -(Wc+Wb+Ws+Wp)          9.62281E+03
 Energy Change up to loading step                              28
    Work Done (elastic unloading)   We         -9.13667E+03
    Stored Strain Energy Change     Wc         -3.53799E+04
    Work Done on Backfill           Wb          2.51151E+02
    Work Done on Support Loads      Ws          0.00000E+00
    Plastic Work Done               Wp          2.31712E+04
    Viscous Work Done               Wv          2.74386E+03
    Excavated Volume                Vm          1.00000E+03
    Strain Energy in Vm             Um          7.58849E+03
    Wk_elastic unload We-(Wc+Wb+Ws+Wp)          2.82086E+03
    Wk_instant unload   -(Wc+Wb+Ws+Wp)          1.19575E+04
Convergence achieved after   28 time steps Error =  23.99 Ferror =  21.85
Mining Step    3 is complete
```

Figure 5-3 Model.OUT File

6.0 MINE2D OUTPUT PROGRAM (OUTPUT)

The OUTPUT program is used to view the geometries of mining steps defined in the input data file "model.INP" and to plot the results calculated in MINE. Start the graphical processor by entering the command OUTPUT. The MAIN MENU will be displayed as shown for the example problem in Figure 7-2. The menu driven OUTPUT program allows the operator to select from a wide variety of result plots and change plot parameters as summarized in Table 6-1.

Main Menu Item	Options/Parameters
PLOT PARAMETERS	colour/line contour plots
COORDINATE RANGE	X-Min, X-Max, Y-Min, Y-Max, Tic Interval
STRESS RANGE	Min, Max, Interval, Units
SAFETY FACTOR RANGE	Min, Max, Interval
STRAIN RANGE	Min, Max, Interval
DISPLACEMENT RANGE	Min, Max, Interval, Units
STRAIN ENERGY RANGE	Min, Max, Interval, Units
STRESS PLOT	S1/n, S2/s, Sxx, Syy, Sxy, Orientation
SAFETY FACTOR PLOT	SF, Failure Zone
STRAIN PLOT	E1/n, E2/s, Exx, Eyy, Exy, Orientation
DISPLACEMENT PLOT	Utotal, Ux, Uy, Orientation
STRAIN ENERGY PLOT	Total, Distortional, Mean
DEFAULT PARAMETERS	(restore defaults for all ranges)
READ RESULT FILE	filename, mining step
VIEW INPUT FILE	filename, mining step
EXIT PROGRAM	

Table 6-1 OUTPUT Program Options

6.1 Main Menu Operation

Menu commands are selected by moving the highlight to the desired command, using the <Up Arrow> and <Down Arrow> keys, and pressing <Enter>.

The first step is to specify the data to be displayed by selecting **VIEW INPUT FILE** or **READ RESULT FILE** and entering the file name and mining step number when prompted at the top of the display.

ie. Select VIEW INPUT FILE from the Main Menu.

```
¦-----File Name (*.inp)----------------------¦
 model.inp <enter>

Mining Step Number?  5 <enter>
```

The geometry of the mining step is displayed as illustrated in Figure 7-3. Zones of non-elastic material (green), open excavations (red), and backfill (yellow) are shown. If a data grid is defined in the model it is outlined in a red dotted line. The area displayed may be controlled as discussed in Section 6.3.

If READ RESULT FILE is selected the model geometry for the mining step, n, is read from "model.INP" and the calculated data is read from "model.00n". The Main Menu commands may then be selected to control plot parameters and request screen plots.

Press the **<Home>** key to return to the Main Menu.

Select **EXIT PROGRAM** from the Main Menu when finished.

6.2 Parameter Editing

When items such as STRESS RANGE are selected from the Main Menu, the current parameters are displayed on a line at the top of the display. Edit the values and press the <Home> key to return to the Main Menu. In cases such as this where parameters may be specified for the plotting of different stress components a function key menu is displayed to select another component or to specify that all components are to use the currently displayed range. STRESS RANGE editing for the example problem is illustrated in Figure 7-8.

6.3 Display Control

The coordinate range defaults are automatically adjusted to show all the zones and grids when an input or result file is first read.

Select COORDINATE RANGE from the Main Menu to display the current plot range parameters. To display a specific area edit the parameters and press <Home> to redisplay and return to the menu.

X-Minimum	X-Maximum	Y-Minimum	Y-Maximum	Tic Int
862.00	1318.00	832.00	1288.00	50.00

The Tic Interval parameter sets the spacing of tic marks around the plot outline.

An alternative method to change to the display area is the use of function keys to zoom in to a smaller area of the existing display.

WINDOW ZOOM may be initiated by pressing the **<Page Down>** key. Use the arrow keys to position the cursor at upper left corner of the window and press **<F1>**. Position the cursor at the lower right corner of the window and press **<F1>**. The display will be redrawn to show the selected area.

Zoom out to SHOW ALL of the model as it appears when first read in by pressing the **<Page Up>** key.

6.4 Calculated Data Plots

Screen plots of the calculated data are requested by selecting the main plot category from the Main Menu. A menu of data components that are available in the category is listed at the top of the display. Enter the selection number of the plot desired. The plots currently available are as follows:

STRESS PLOTS	
S1/n	contours of maximum principal stress (normal on faults)
S2/s	contours of minimum principal stress (shear on faults)
Sxx	contours of normal stress component in the X-axis direction
Syy	contours of normal stress component in the Y-axis direction
Sxy	contours of shear stress component
Orientation	major and minor principal stress axes
SAFETY FACTOR PLOTS	
SF	contours of safety factor
Failure Zone	color map of plastic yielded zones
STRAIN PLOTS	
E1/n	contours of maximum principal strain (normal on faults)
E2/s	contours of minimum principal strain (shear on faults)
Exx	contours of strain component in the X-axis direction
Eyy	contours of strain component in the Y-axis direction
Exy	contours of shear strain component
Orientation	major and minor principal strain axes
DISPLACEMENT PLOTS	
Utotal	contours of total displacement
Ux	contours of displacement in the X-axis direction
Uy	contours of displacement in the Y-axis direction
Orientation	displacement orientation
STRAIN ENERGY DENSITY PLOTS	
(see Jaeger & Cook, 1969, Section 5.8)	
Total	contours of total stored strain energy density
Distortional	contours of distortional strain energy component
Mean	contours of mean strain energy component

Table 6-2 Calculated Data Plots Available in OUTPUT

During the contouring process, any results within type 2, type 3, and type 4 zones will be plotted on top of any grid results.

Contour Limits

The default contour ranges for calculated data plots are read from a file named **"MINE2D.PAR"**. This file is written in ASCII format and may be edited to change the defaults to ranges more suited to the problems being run or to change the system of units. The ranges may be temporarily edited as described in Section 6.2.

In the contour legend, the bottom and top colors indicate underflow and overflow conditions. Data values outside of the specified range will be plotted in one of these colors. Specified contour intervals are represented by the transition from one color to the next, with the minimum and maximum values occurring at the edge of the underflow and overflow colors. In the SAFETY FACTOR plot, a special color added at the bottom of the legend, identifies areas of failure in tension.

6.5 Function Keys

F1 or **<Home>**	Returns control to the MAIN MENU.
F7 or **<Page Up>**	Redisplays the mining step geometry using the default coordinate range to SHOW ALL zones and grids in the model.
F8 or **<Page Down>**	Begins WINDOW ZOOM by placing a cross-hair cursor on the screen. Use the arrow keys to move the cursor to one corner of the new viewing window and press **F1**. Move cursor to the opposite corner and press **F1** again.
<Print Screen>	Copies screen display to the printer or a file if third party software such as Pizazz Plus has been installed and activated.

7.0 EXAMPLE PROBLEM

In this example a simple sequence of excavation and backfilling is modelled to demonstrate the preparation of MINE2D models, running of the computational analysis, and the generation of result plots.

The first step is to prepare a layout defining the model geometry as shown in Figure 7-1. A sequence of vertical retreat mining and backfilling is to be modelled in a horizontal plan. The orebody has been divided into zones that correspond to the planned and possible mining panels. Type 4 is used for Zone 5 which is excavated in the first mining step and never filled. The use of type 3 for remaining ore zones permits the assignment of alternate non-elastic material properties and the placement of backfill in excavated panels. The ore panels include the pillars and abutments where failure is most likely to occur in the mining sequence that is to be modelled. A fault is represented in the model by type 2 zones. As described in Section 4.2.3, adjacent zones must share common corner points and must have an equal number of subdivisions along common edges. The number of subdivisions is set to 7 per side in all zones. The data grid is located to provide results outside the defined zones.

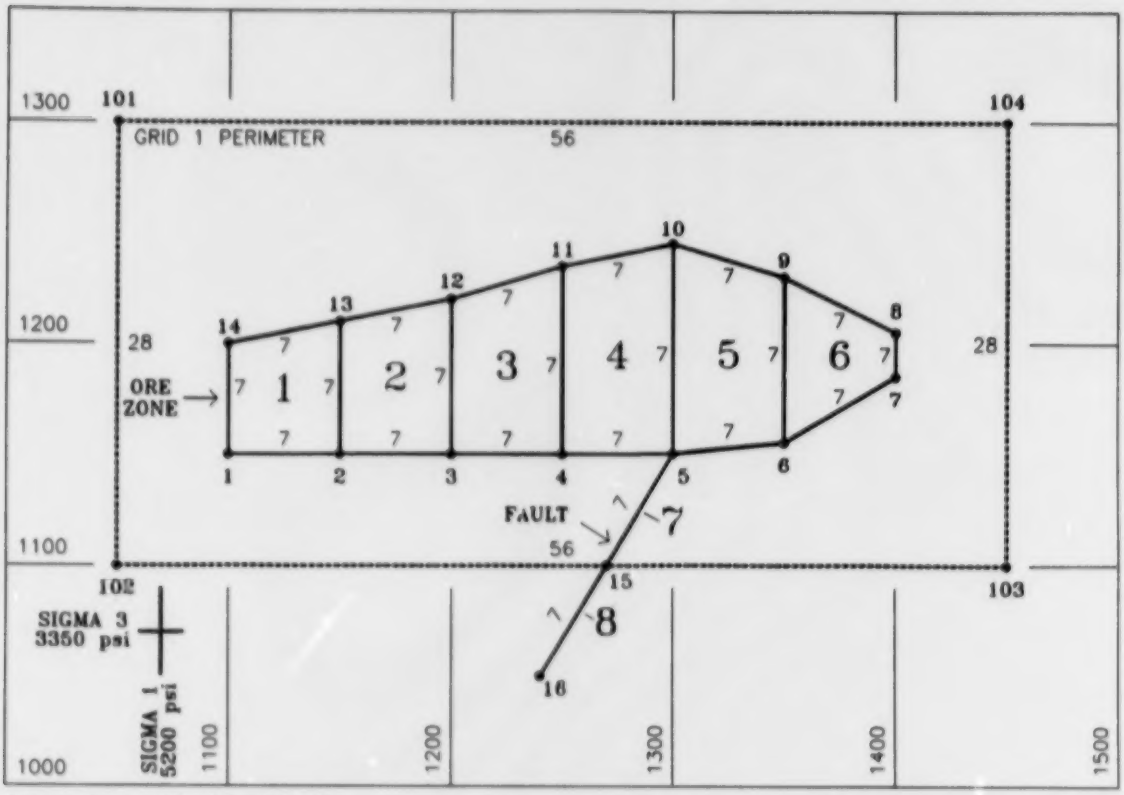
Input File Preparation

To provide a guideline for the preparation of the input data file copy "SAMPLE.INP" to "EXAMPLE1.INP". This file must be modified using an editor or word processor so that the 8 parameter sets define the desired model geometry, parameters and mining sequence according to the format described in Section 4.2. The file contains many comment lines which begin with an "*" in the first column. Comments may be added or deleted but no blank lines or other unexpected lines are permitted. Parameter sets that use more than one line are terminated by "0/" on the last line.

The JOB DESCRIPTION to be used on plots is input as a single line.

```
*  ------------------------------------------------------------------------------
* JOB DESCRIPTION SECTION ( used in output plot headings )----|
  SAMPLE MINE2D MODEL
*  ------------------------------------------------------------------------------
* CONTROL PARAMETERS SECTION
* |-----------|-----------|---------------------|----------|----------|
* |Max. # of  |Max. # of  |Relaxation Parameter  | Stress   | Symmetry |
* |Load Steps |Iterations |  Maximum  |  Minimum |Tolerance | Planes   |
* |/Mng. Step |/Load Step |           |          |(psi/MPa) |          |
* |-----------|-----------|-----------|----------|----------|----------|
        1000        1000       1.200      0.200    5.0000          0
*  ------------------------------------------------------------------------------
```

In the CONTROL PARAMETERS SECTION the Stress Tolerance is set at 5 psi (approx. 0.1% of the field stress) and Symmetry Planes is zero. The other control parameters are left at the recommended values.



Mining Step 1 - assign ore and fault material properties
- excavate Zone 5

Mining Step 2 - excavate Zone 2

Mining Step 3 - pressurize Zone 2 to initial fill pressure

Mining Step 4 - place backfill in Zone 2 at 100 psi

Mining Step 5 - excavate Zone 3

Figure 7-1. Layout for EXAMPLE1 Model

In the ZONE GEOMETRY DEFINITION LIST each of the zones shown in Figure 7-1, are identified by consecutive zone numbers. For the type 3 and type 4 zones the coordinate numbers of the corner points are listed in a counterclockwise direction. The number of elements that the zones will be divided into for the analysis is defined by specifying the number of divisions along the edges from Pt. 1 to Pt. 2 and from Pt. 2 to Pt. 3. The type 2 zones used to represent faults only require two end points. Each zone is divided into 7 elements. The thickness is set to 0 because no elastic gouge material is to be modelled.

ZONE GEOMETRY DEFINITION LIST

Zone Num	Coordinate Number Pt. 1	Pt. 2	Pt. 3	Pt. 4	# of Div Between 1-2	2-3	Type	Thickness (type 2)	Description (optional)
1	1	2	13	14	7	7	3	0.00	
2	2	3	12	13	7	7	3	0.00	
3	3	4	11	12	7	7	3	0.00	
4	4	5	10	11	7	7	3	0.00	
5	5	6	9	10	7	7	4	0.00	
6	6	7	8	9	7	7	3	0.00	
7	5	15	0	0	7	0	2	0.00	'fault'
8	15	16	0	0	7	0	2	0.00	'fault'

0/

COORDINATE LIST

Coord Number	X Coord	Y Coord
1	1100.00	1150.00
2	1150.00	1150.00
	etc.	

0/

The COORDINATE LIST must contain the coordinates of all points referred to in the ZONE GEOMETRY and GRID DEFINITION LISTS.

In the STRESS STATES section, the far field stress state is input for material 1, the host rock mass. In this horizontal plan model the stresses do not vary with the Y coordinate. The desired stress magnitudes of 5200 psi and 3350 psi are assigned to the b component of Sigma:1 and Sigma:2 respectively. Theta is set to 90 degrees to specify that the Sigma:1 direction is 90 degrees counterclockwise from the X-axis. The far field stress values are also input as the initial stress state on the ore, material 2, and on the fault zone, material 3. An initial hydrostatic stress state of 100 psi is input for the backfill, material 4. These compressive stresses are input as positive values in MINE2D.

```
* ------------------------------------------------------------------------------
* STRESS STATES
* |---------|-----------------------------------------|---------|
* |Material |          Principal Stresses             |         |
* | Number  |Sigma:1 = a * y + b|Sigma:2 = a * y + b|  Theta  |
* |---------|----a----|----b----|----a----|----b----|---------|
* ie. far field stresses in the host material
          1      0.00   5200.00      0.00   3350.00     90.00
* ie. initial field stresses in ore
          2      0.00   5200.00      0.00   3350.00     90.00
* ie. initial field stresses on fault
          3      0.00   5200.00      0.00   3350.00     90.00
* ie. initial stresses in backfill material
          4      0.00    100.00      0.00    100.00      0.00
          0/
* ------------------------------------------------------------------------------
* MATERIAL PROPERTIES LIST
* |---------|---- Peak Strength/Residual Strength/G1 - G2 ----------------|
* |Material |  Young's  |Poisson's |    To    |    Co    |   So,m   |  phi,s   |
* |Num |Type|Modulus/G1| Ratio/G2 |          |          |          |          |
* |----|----|----------|----------|----------|----------|----------|----------|
* ie. host material
     1    1 1.0000E+07        0.30   -300.00   9000.00      0.00     45.00
            0.0                0.00      0.00      0.00      0.00      0.00
            0.0                0.00      0.00      0.00      0.00      0.00
* ie. material properties for ore
     2    1 0.7200E+07        0.30   -300.00   8000.00      0.00     35.00
            0.0                0.00      0.00      0.00      0.00      0.00
            0.7200E+06         0.00      0.00      0.00      0.00      0.00
* ie. properties to be assigned to the fault
     3   -1 0.00              0.00      0.00  99999.00      0.00     30.00
            0.00               0.00      0.00      0.00      0.00      0.00
            0.7200E+06         0.00      0.00      0.00      0.00      0.00
* ie. high density cemented backfill
     4    1 2.9000E+04        0.25    -25.00    400.00      0.00     24.00
            0.0                0.00      0.00      0.00      0.00      0.00
            2.9000E+03         0.00      0.00      0.00      0.00      0.00
   0/
* ------------------------------------------------------------------------------
```

In the MATERIAL PROPERTIES LIST material number 1 is reserved for the elastic host rock mass. The choice of Material type 1 for the host indicates that Mohr-Coulomb failure criteria are to be used to calculate safety factors at field grid points. Materials 2 and 4, define the properties of ore and backfill respectively. Material type 1 indicates that the ore and backfill placed in type 3 zones will behave in an elasto-visco-plastic manner and that Mohr-Coulomb failure criteria will be used. For all three materials the elastic properties; Young's Modulus and Poisson's Ratio, are input on the first of three data lines. Likewise, the Mohr-Coulomb failure criteria parameters; To (tensile strength), Co (field scale compressive strength), and phi (friction angle) must be specified.

Material 3, is used to define the properties that will be assigned to the fault (type 2 zones). Material type -1 indicates that the fault is allowed to behave in a rigid visco-plastic manner only, i.e. fault slip but no elastic response. For the Mohr-Coulomb failure criteria; To and So (cohesion) are set to zero, Co is set

very high to disable any compressive failure, and the angle of friction (phi) is set to 30 degrees.

As a first guess the viscous parameters for materials 2 and 4 are set at G1 equal to 1/10th of the Young's Modulus and G2 equal to zero. Material number 3 is assigned the same viscous parameters as material number 2.

In the GRID DEFINITION LIST the field point grid is defined by listing the four corners in a counterclockwise direction, specifying the number of divisions along the first two sides, and selecting material number 1 strength criteria for the calculation of safety factors.

```
* ------------------------------------------------------------------------------
* GRID DEFINITION LIST
* |-------|---------------------------|-------------|------------|
* |  Grid |      Coordinate Number    | # of Div    |            |
* | Number| Pt.  | Pt.  | Pt.  | Pt.  |Between Pt.  |  Material  |
* |       | One  | Two  |Three | Four | 1-2  | 2-3  |  Number    |
* |-------|------|------|------|------|------|------|------------|
        1    101    102    103    104     28     56       1
       0/
* ------------------------------------------------------------------------------
```

In the MINING STEPS section the sequence outlined in Figure 7-1 is defined by assigning materials or stress states to the previously defined zones. In the first step alternate material properties are assigned. Properties of the ore, material 2 and stress state 2, are assigned to type 3 zones by specifying the zone number followed by the material code 2 or +2. Similarly, the properties of material 3 are assigned to the type 2 zones which represent the fault. Excavating zone 5 to zero surface stress is specified by inputting the zone number and a 0 code. A "0/" line is used to mark the end of each mining step. In Mining Step 3 the previously excavated zone 2 is prestressed to the initial stress state of the backfill by inputting the negative or the stress state number. In Mining Step 4 the backfill material is placed in zone 2. Zone 3 is excavated to zero stress in Mining Step 5 to test the behaviour of the backfill and of the pillar.

```
* ------------------------------------------------------------------------------
* MINING STEPS
* |---------|---------|
* |  Zone   |         |
* | (block) |  Code   |
* |---------|---------|
* Mining Step    1
* assign material 2 properties to the ore zones except zone 5 (type 4)
            1         +2
            2         +2
            3         +2
            4         +2
            6         +2
* assign material 3 properties to the fault
            7         +3
```

```
          8        +3
* excavate panel 5 which will be left unfilled
          5         0
          0/
* Mining Step    2    excavate panel 2
          2         0
          0/
* Mining Step    3    prestress panel 2 to 100 psi
          2        -4
          0/
* Mining Step    4    place consolidated fill in panel 2 at 100 psi
          2        +4
          0/
* Mining Step    5    excavate panel 3
          3         0
          0/
* ------------------------------------------------------------------------------
```

Checking the Input

The completed input data file may be checked by running OUTPUT to view the geometry of each mining step. When OUTPUT is run the initial display contains the Main Menu shown in Figure 7-2. Press the <Down Arrow> key to move the highlight to VIEW INPUT FILE and press <enter>. Enter the file name "example1.inp" and the mining step number in response to the prompts at the top of the screen. The selected mining step is displayed as illustrated in Figure 7-3 for Mining Step 5. The backfilled panel is shown in yellow and the open excavations are shaded red. A dotted red outline shows the location of the field point grid.

Running the Analysis

The computational analysis is begun by reading "example1.inp" into MINE as described in Section 5.1 . This problem contains 441 elements. It can be run using the 675 control point version of MINE. On a 25 Mhz. 386 with a Wietek coprocessor and 16 Mb. of RAM the five step analysis requires about 5 hours. The progress of the analysis may be monitored from reports of the solution parameters on the display. The energy release data calculated for each mining step is printed to the file "example1.out". The result data files written for OUTPUT are example1.001, example1.002, example1.003, example1.004, and example1.005 . In addition file "example1.txt" contains details of the basic calculated data in ASCII format.

Generating Data Plots

The OUTPUT program is run to generate plots of the calculated data. Select READ RESULTS FILE from the Main Menu. Enter the file name "example1" and mining step number "5" to read in the results for step 5. Both the "example1.inp" and "example1.005" files will be read. The geometry of mining step 5 is displayed as it was for viewing the input file in Figure 7-3. The default coordinate range is calculated to show all zones and grids in the model.

To change the area that is displayed select COORDINATE RANGE from the Main Menu. Edit the coordinate range parameters as shown at the top of the display in Figure 7-4. Press <HOME> to display the selected area and return to the Main Menu as shown in Figure 7-5.

Screen plots of the calculated data are requested by selecting the plot category from the Main Menu and then selecting the component that is to be plotted from the prompt at the top of the screen. A detailed list of plot types is provided in Section 6.4.

Select STRESS PLOT from the Main Menu to display the options at the top of the screen and type 1 to display the plot of Sigma:1 shown in Figure 7-6.

Stress: S1, S2, Sxx, Syy, Sxy, Orientation (1,2,3,4,5,6) ? 1

The areas shown in bright red or in grey indicate that the stress magnitudes are above or below the current stress contouring range. The color coding on the fault indicates the magnitude of the normal stress on the fault.

If a Print Screen utility has been installed press <Print Screen> to print a copy of the plot. The color plots in this manual have been printed using Pizzaz Plus and a HP Paintjet printer.

Press <Home> to return to the Main Menu.

In addition to color contoured plots of data magnitude, orientation plots may be requested such as that for the principal stress axes in Figure 7-7.

To examine the stress distribution in the backfill change the coordinate range to zoom into the area around panel 2. Select STRESS RANGE to edit the contouring range, as shown in Figure 7-8 and replot Sigma:1. The plot in Figure 7-9 shows Sigma:1 values in the backfill are increased locally to about 160 psi.

Some of the other plots of stress, safety factor, failure zone (yield), strain, displacement, and strain energy density which may be selected are illustrated in Figures 7-9 through 7-19.

Figure 7-12 shows that in the plastic zones the Safety Factor does not go below 1 because of the yielding which is permitted as shown in the Failure Zone plot, Figure 7-13.

An input data file may be prepared to restart the MINE2D analysis and continue the mining sequence or test alternatives as described in Section 4.2.9. Figures 7-20 and 7-21 show Sigma:1 and Total Displacement after the mining sequence has been continued to fill panel 3 and remove the pillar, panel 5.

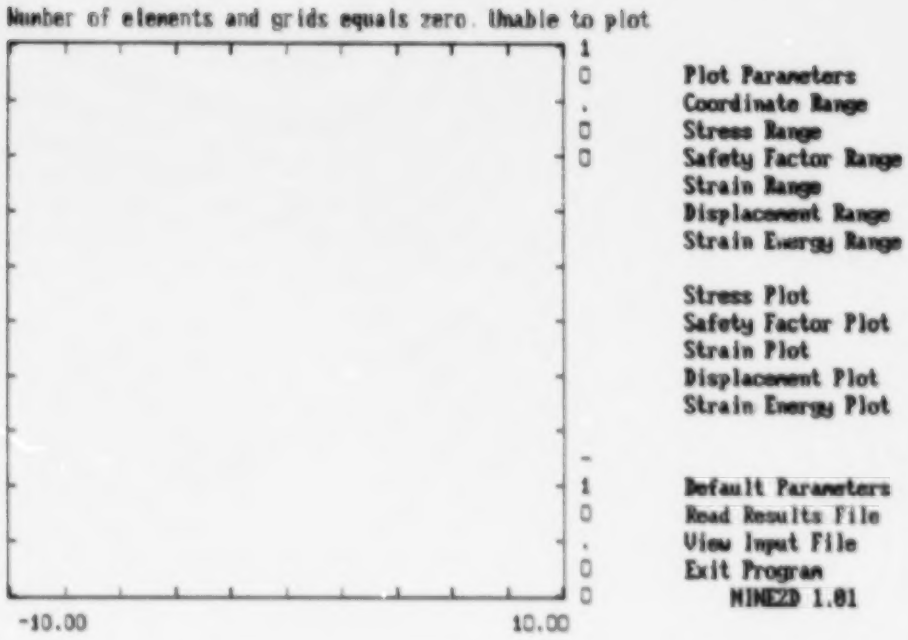


Figure 7-2 Initial Screen Display of the Main Menu in OUTPUT

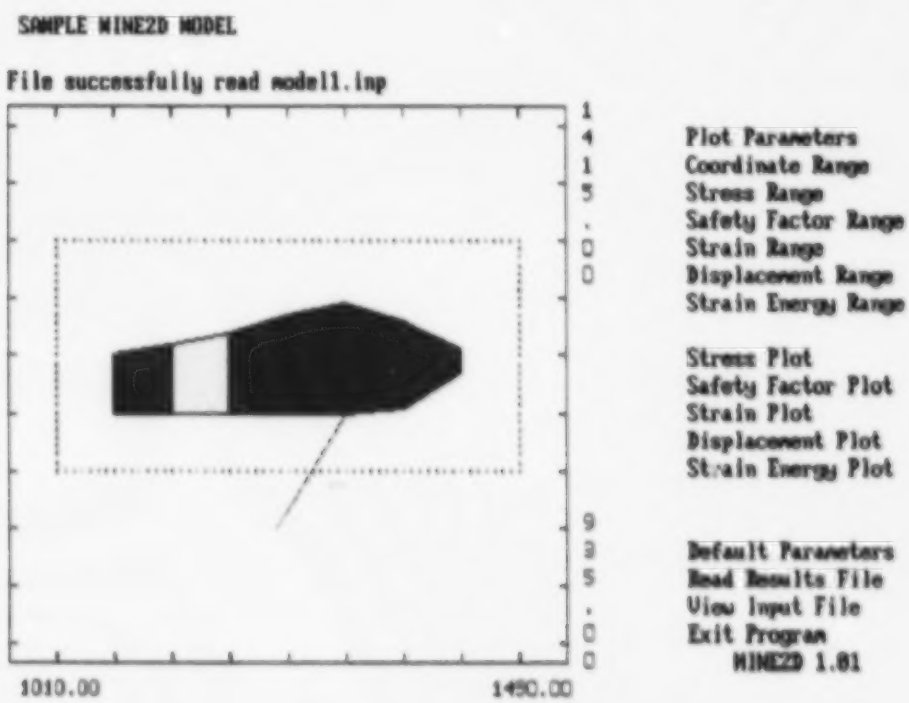


Figure 7-3 Using VIEW INPUT FILE to Check Mining Step 5

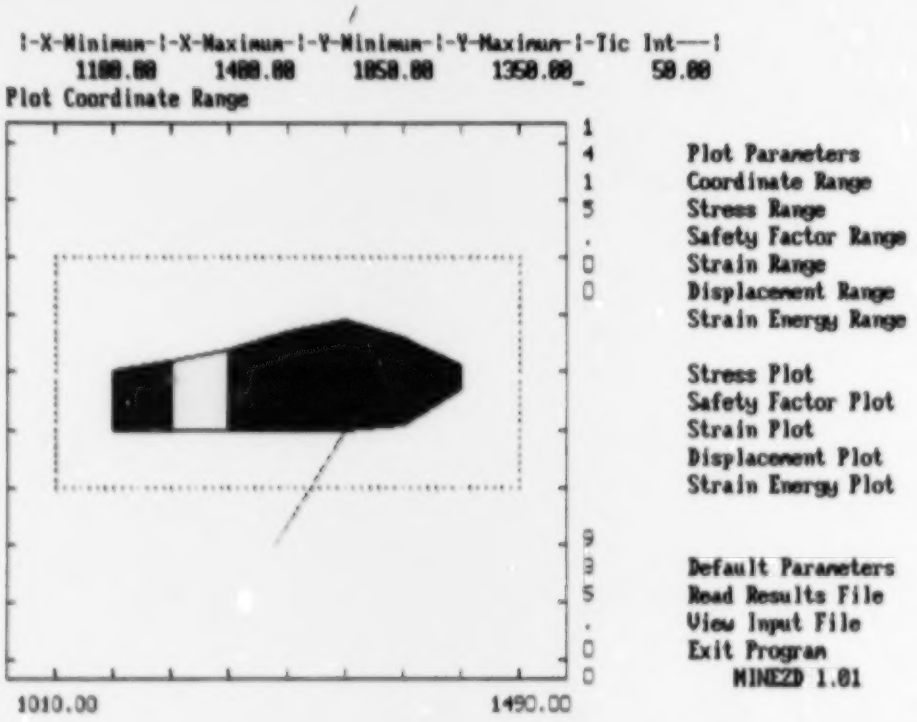


Figure 7-4 Editing the Plot COORDINATE RANGE Parameters

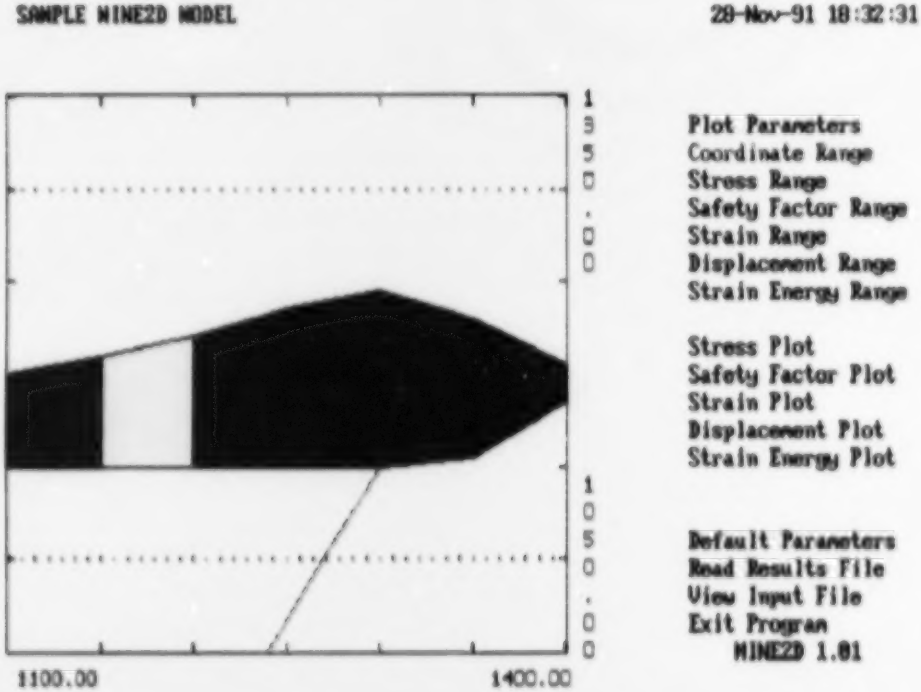


Figure 7-5 Mining Step Redisplayed to Show Selected Area

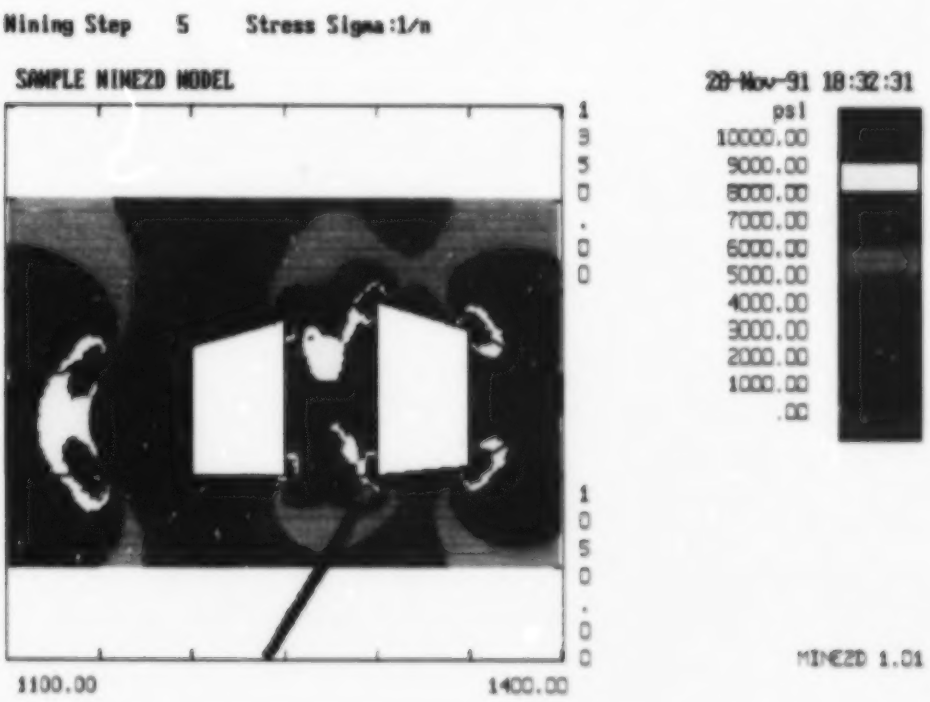


Figure 7-6 Sigma:1 Stress Contour Plot with Sigma:n on Fault

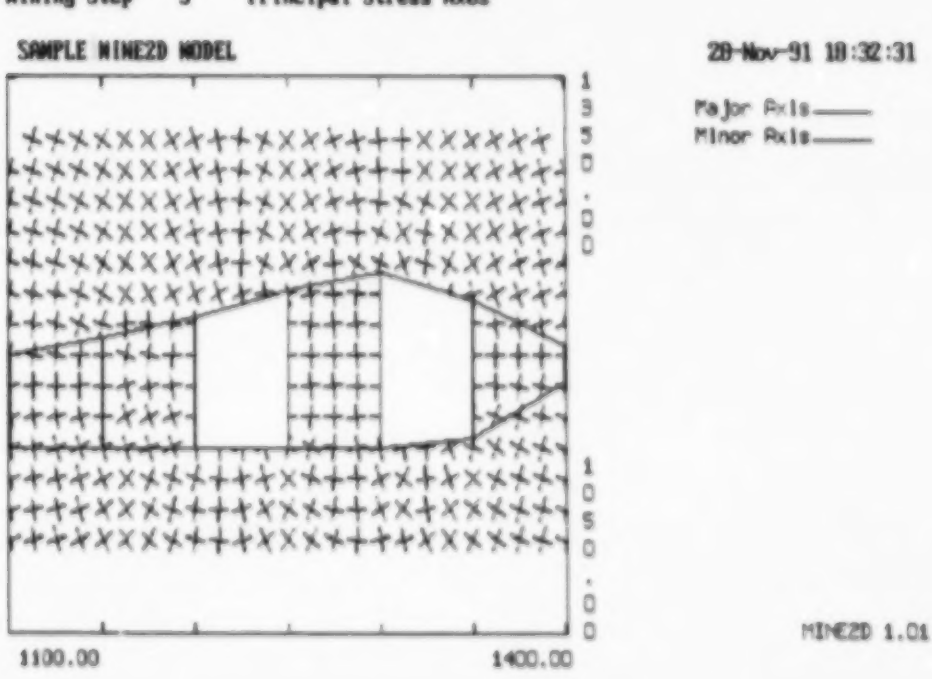


Figure 7-7 Principal Stress Axes Orientation Plot

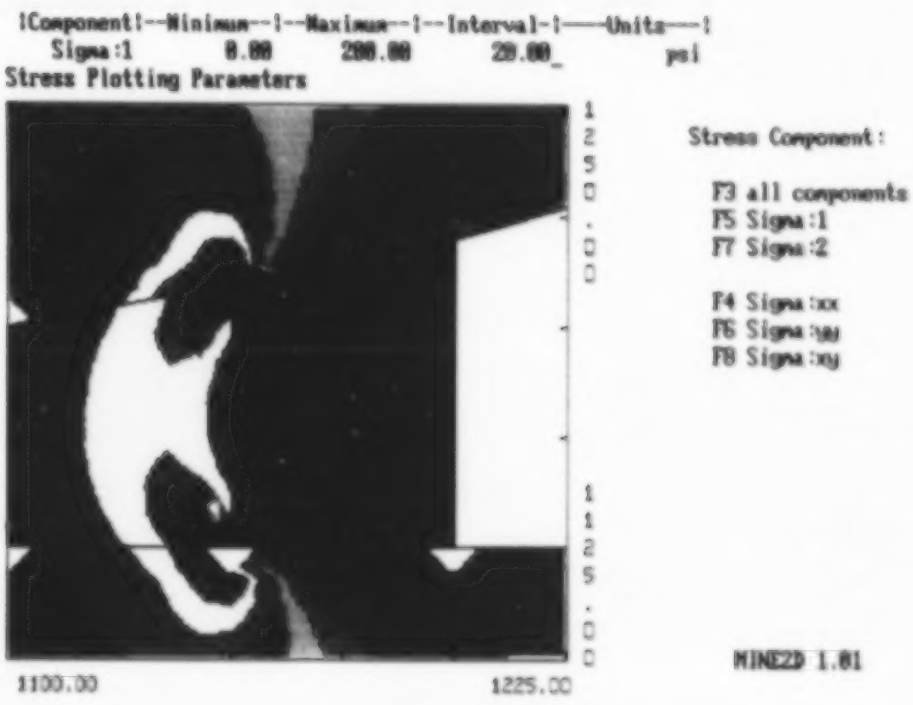


Figure 7-8 Editing the STRESS RANGE Parameters

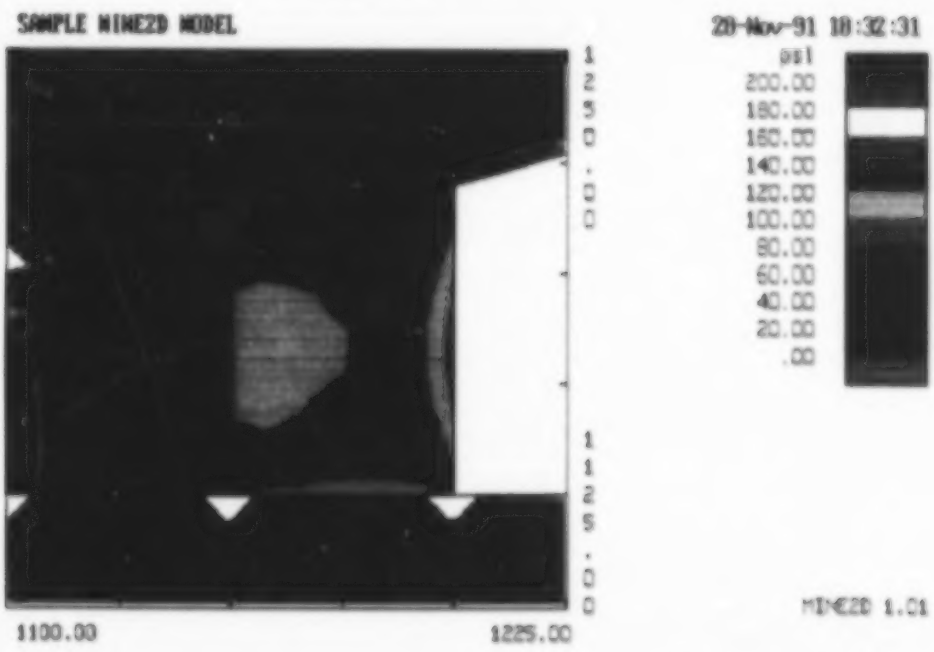


Figure 7-9 Sigma:1 Contoured to Show Stress in Backfill

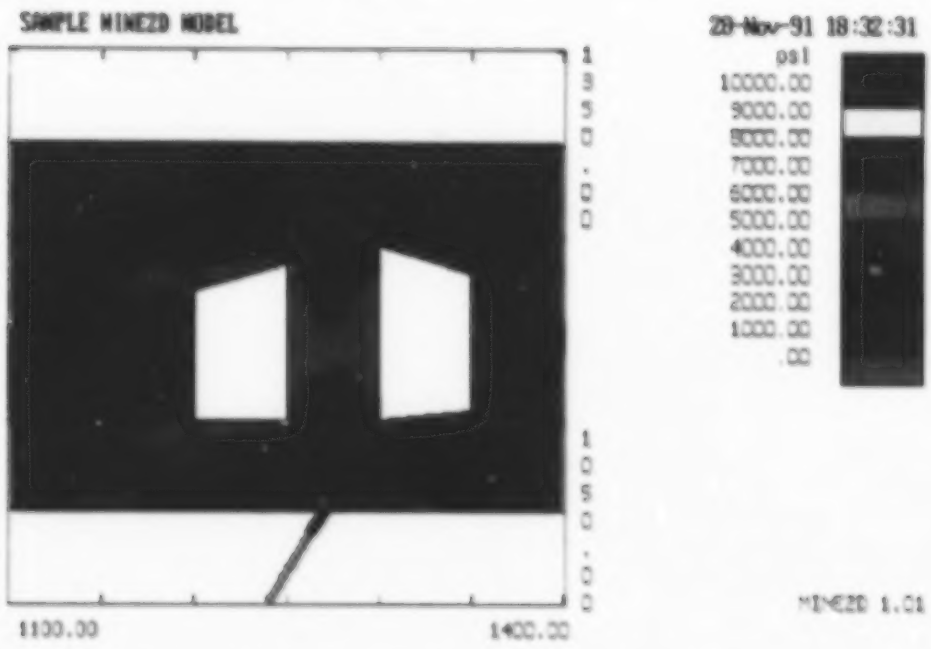


Figure 7-10 Sigma:2 Stress Contour Plot with Sigma:s on Fault

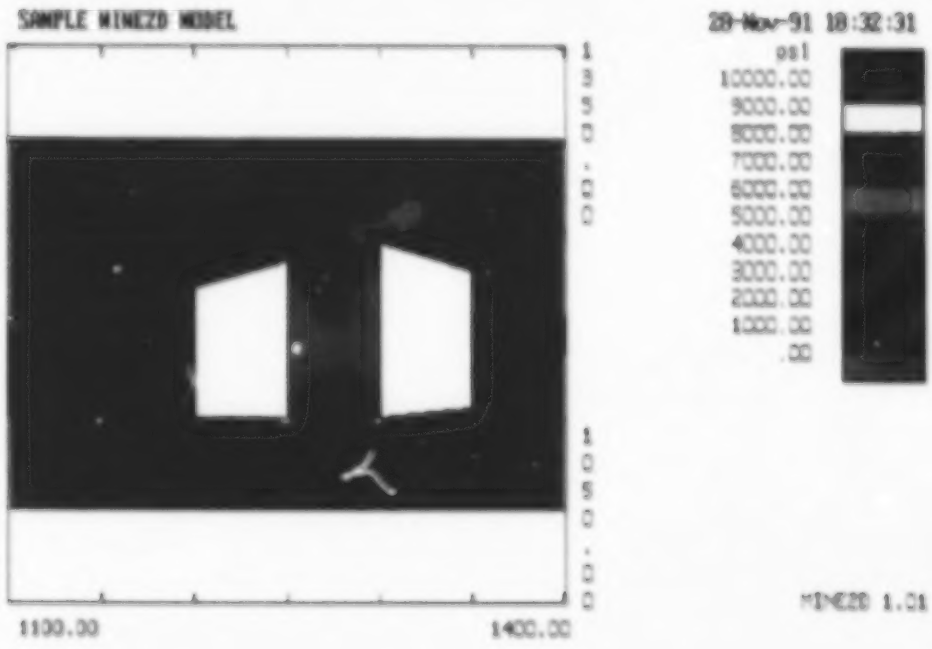


Figure 7-11 Stress in the X-Axis Direction

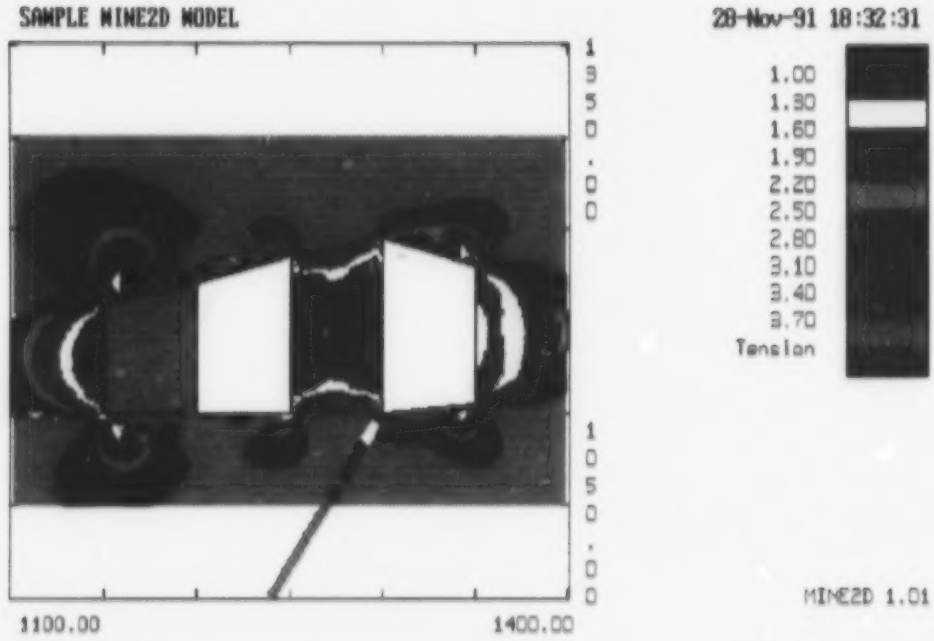


Figure 7-12 Safety Factor Contour Plot

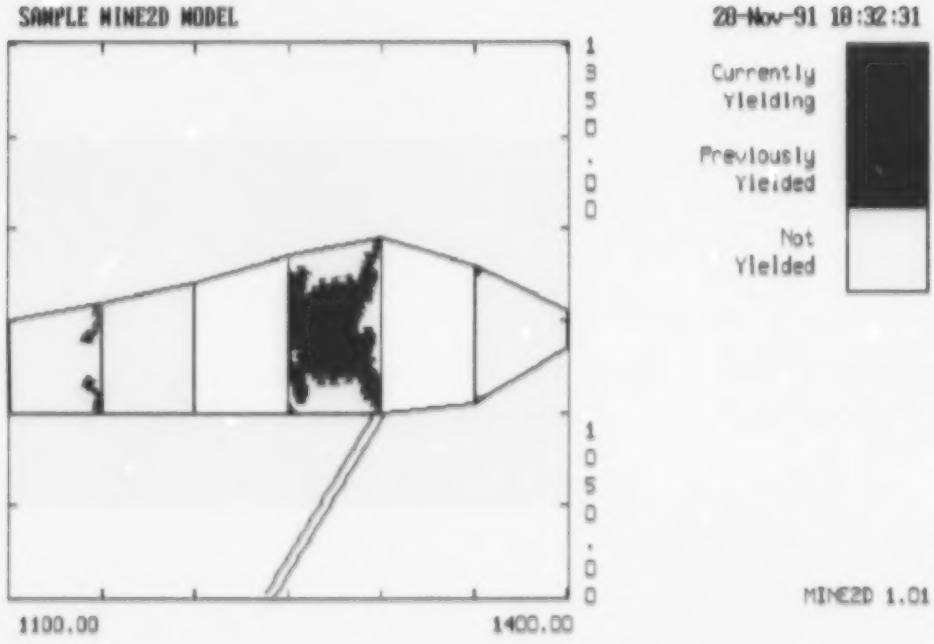


Figure 7-13 Failure Zone Plot of Plastic Yielding Areas (Elastic only material not allowed yield)

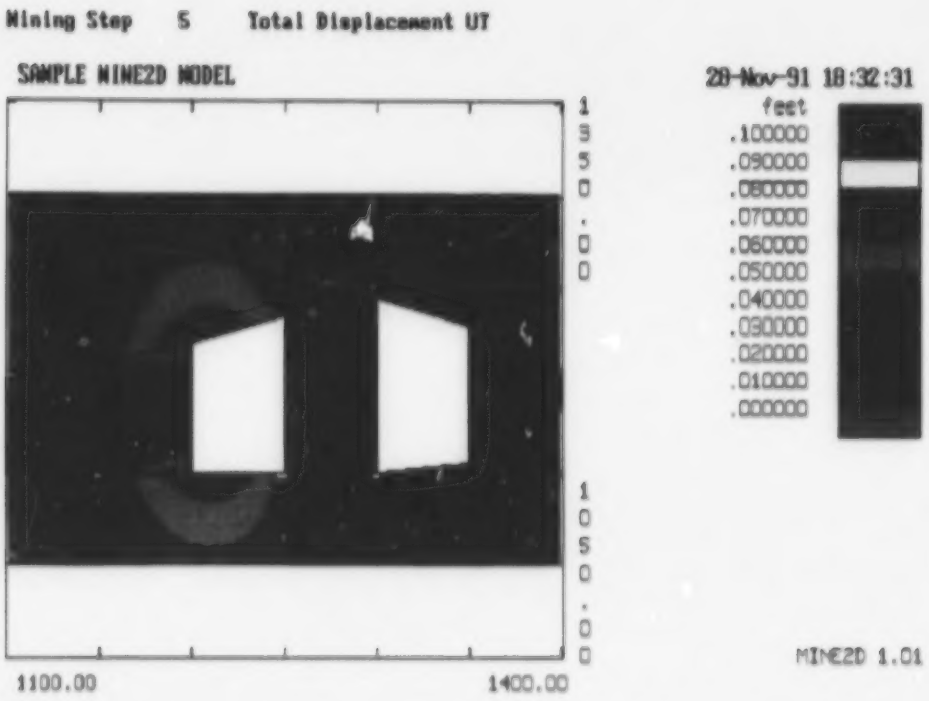


Figure 7-14 Total Displacement Contour Plot

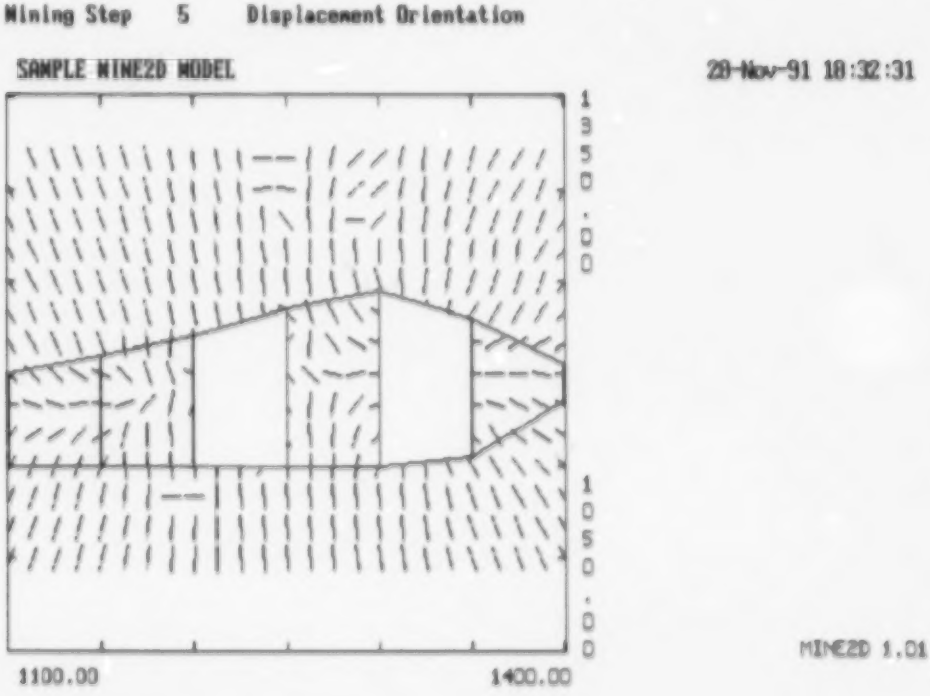


Figure 7-15 Displacement Orientation Plot

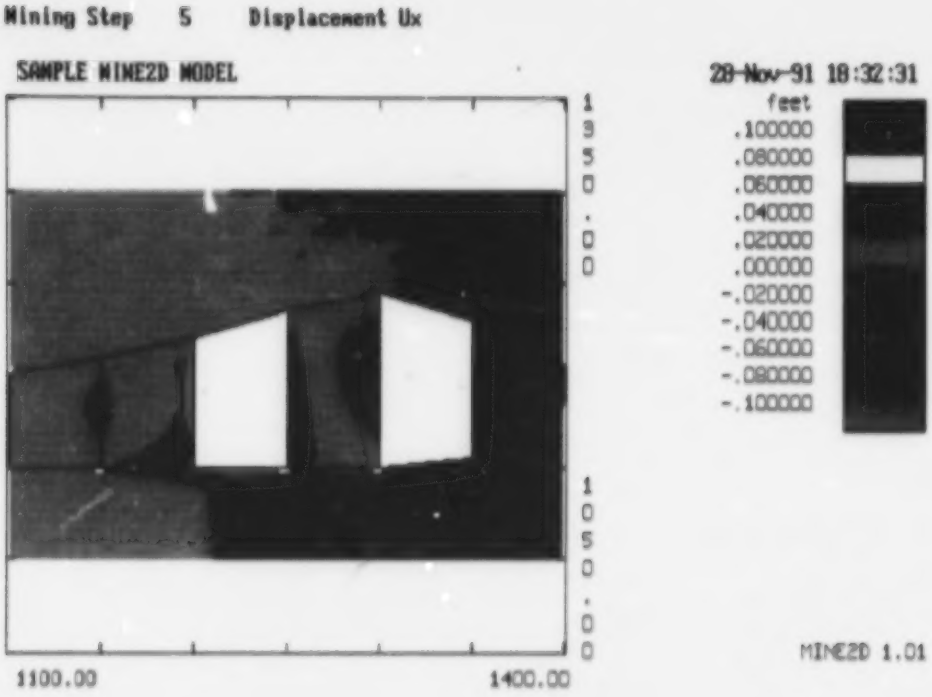


Figure 7-16 Displacement in the X-Axis Direction

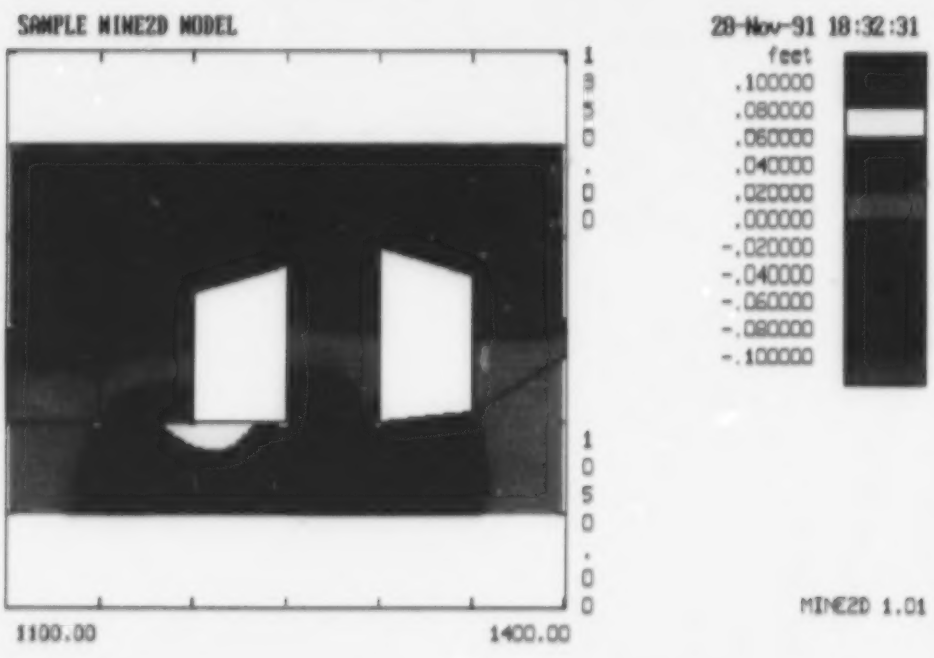


Figure 7-17 Displacement in the X-Axis Direction

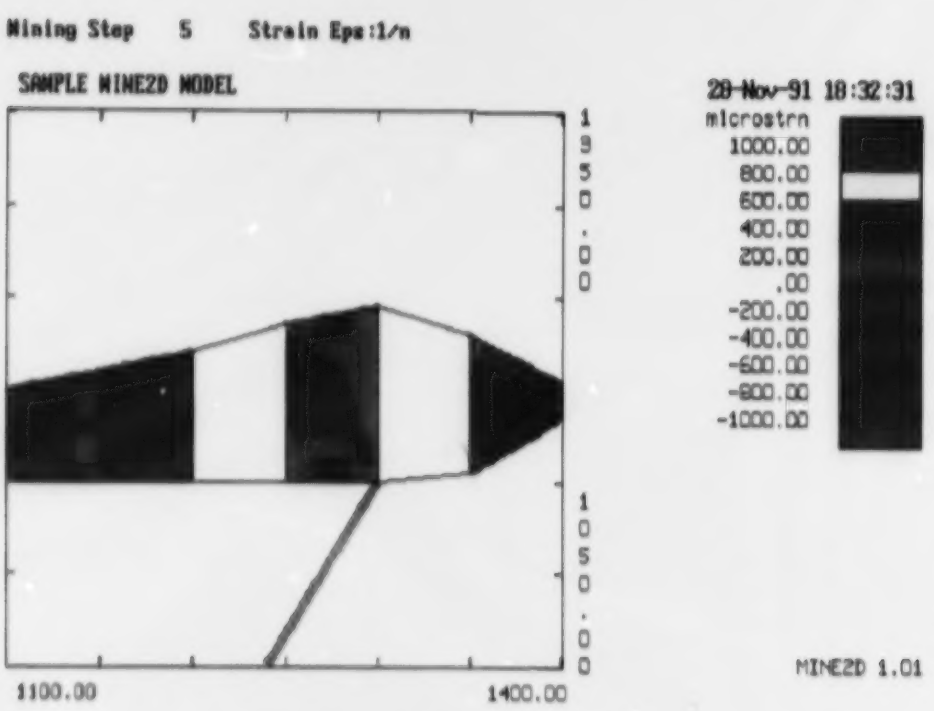


Figure 7-18 Eps:1 Plastic Strain Plot with Eps:n on Fault

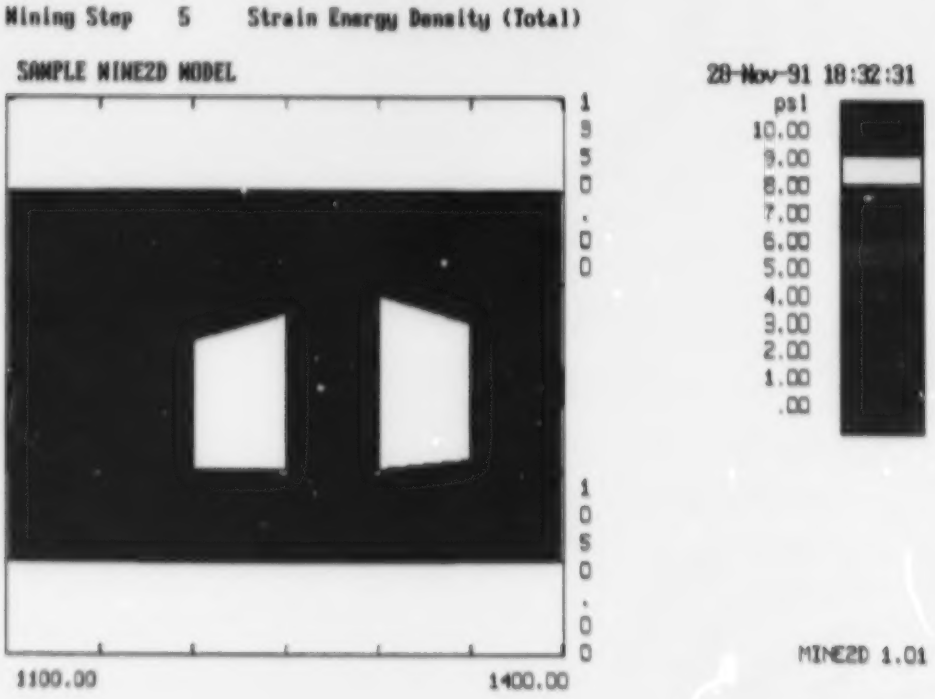


Figure 7-19 Stored Elastic Strain Energy Density Plot

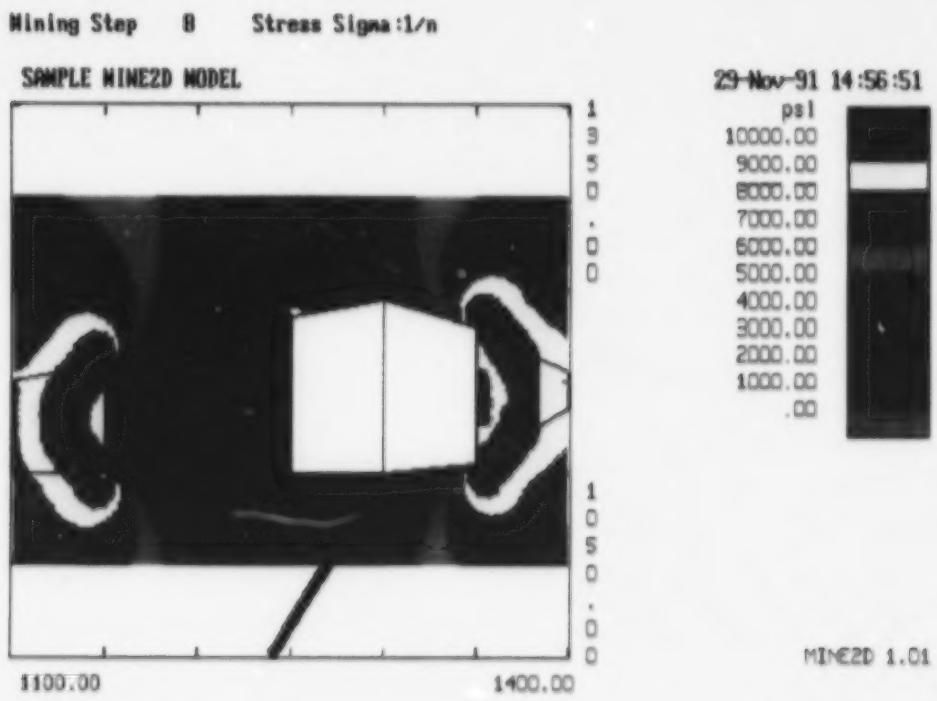


Figure 7-20 Sigma:1/n Stress Plot after Mining Pillar

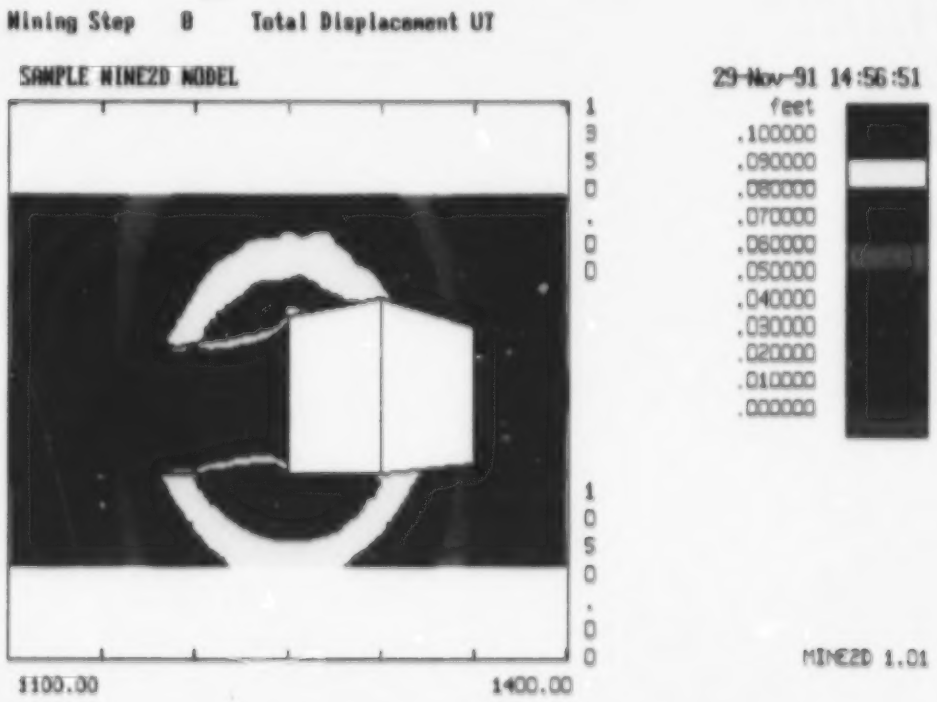


Figure 7-21 Total Displacement Plot after Mining Pillar

8.0 REFERENCES

Banerjee, P.K. and Butterfield, R., 1981. Boundary Element Methods in Engineering Science, McGraw-Hill Book Company (UK) Limited, London

Crouch, S., 1976. "Analysis of stresses and displacements around underground excavations: an application of the displacement discontinuity method:, Dept. Civil Engineering., U. of Minnesota, Minneapolis, Minnesota, U.S.A.

Hoek, E. and Brown, E.T. 1988. The Hoek-Brown failure criterion - a 1988 update. Proc. 15th Canadian Rock Mech. Symp., University of Toronto. p. 31-38.

Jaeger, J.C. and Cook, N.G.W., 1976. Fundamentals of Rock Mechanics, Chapman and Hall, a Halsted Press Book, John Wiley & Sons Inc., New York, New York, U.S.A.

Wiles, T. and P. MacDonald. 1990. Rock mechanics aspects of regional faulting around Creighton mine. Rock Mechanics Contributions and Challenges: Proceedings of the 31st U.S. Symposium, Golden, Colorado, p. 777-784.

Wiles, T. 1989a. Non-linear modelling of mining at great depth: a case history at Creighton mine. Int. Symp. Rock Mechanics, Pau, France, V.2, p. 621-628.

Volumes of the Final Report

on

Simulation of Bulk Mining at Depth with Backfill in Ontario Mines

Volume 1	EXECUTIVE SUMMARY
Volume 2	MATH. FORMULATION AND CODE DEVELOPMENT
Volume 3	FIELD MONITORING PROGRAM
Volume 4	NUMERICAL MODEL CALIBRATION STUDIES
Volume 5	MINE2D USER'S MANUAL
Volume 6	MINE3D USER'S MANUAL

APPENDIX I

"SAMPLE.INP" FILE LISTING

```
*  ------------------------------------------------------------------------------
* INPUT FILE FOR MINE2D NUMERICAL MODEL
* Comment lines begin with * and may be edited, added or removed.
* Each data section must be present in the order presented here.
* Each line must contain the number of values shown seperated by spaces or ,
*  ------------------------------------------------------------------------------
* JOB DESCRIPTION SECTION ( used in output plot headings )
*|--------------- one line only, 60 char. max. -----------------|
  SAMPLE MINE2D MODEL
*  ------------------------------------------------------------------------------
* CONTROL PARAMETERS SECTION
* Symmetry  = 0 for no symmetry
*           = 1 for reflection such that x <=> -x
*           = 2 for reflection such that x <=> -x and y <=> -y
* If symmetry is used the coordinate system used to describe the model
* geometry must be located so that the axes of symmetry are at y=0 and x=0.
* |-----------|-----------|----------------------|-----------|-----------|
* |Max. # of  |Max. # of  |Relaxation Parameter  | Stress    | Symmetry  |
* |Load Steps |Iterations |  Maximum  |  Minimum |  Tolerance|  Planes   |
* |/Mng. Step |/Load Step |           |          | (psi/MPa) |           |
* |-----------|-----------|-----------|----------|-----------|-----------|
       1000        1000       1.200       0.200     5.0000          0
*  ------------------------------------------------------------------------------
* MODEL GEOMETRY DEFINITION LIST
*
* Model geometry is defined by type 1 or type 2 line segments, or
* type 3 or type 4 zones.
*
* Types 1 and 2 are input by specifying the two end point coordinates.
*
* Types 3 and 4 are input by specifying the four corner point coordinates in a
* counterclockwise sense.
*
* For types 3 and 4 a triangular zone may be defined by assigning two adjacent
* points the same coordinate number. The # of divisions on sides 1-2 and 2-3
* must be the same for triangular zones.
*
* Zones may not cross and where they join along common edges must have the same
* end point coordinate numbers and number of divisions.
*
* Dummy values (ie. 0) required where numeric fields are not used for a zone.
*
* ZONE TYPES
*   1 Single segment of FICTITIOUS FORCE (Surface) elements
*     try to avoid these elements - complex rules apply
*     (specify Pt.1 & Pt.2 & Div 1-2)
*   2 Single segment of DISPLACEMENT DISCONTINUITY elements
*     ie. faults, joints, seams
*     (specify Pt.1 & Pt.2 & Div 1-2 & thickness)
*   3 Three or four sided zone of INITIAL STRESS (Volume) elements
*     ie. materials other than host, excavations to be filled, zones with
*         different pre-mining stress, zones of non-elastic material
*     (specify Pt.1 & Pt.2 & Pt.3 & Pt.4 & Div 1-2 & Div 2-3)
```

```
*   4 Three or four sided zone bounded by FICTITIOUS FORCE (Surface) elements
*     ie. as a construction convenience zones of host material may be excavated
*         and forces applied to the faces but may not be filled
*     (specify Pt.1 & Pt.2 & Pt.3 & Pt.4 & Div 1-2 & Div 2-3)
*
* Use a 0/ zone number to end the list
*
*|----|--------------------|---------|----|---------|-------------------------|
* |Zone| Coordinate Number  | # of Div|    |Thickness|                         |
* | Num| Pt.| Pt.| Pt.| Pt.| Between |Type|( type 2)|       Description       |
* |    |  1 |  2 |  3 |  4 | 1-2| 2-3|    |         |        (optional)       |
*|----|----|----|----|----|----|----|----|---------|-------------------------|
* ie. ore blocks
      1    5    6   11   12    5    5    3       0.00
      2    6    7   10   11    5    5    4       0.00
* ie.fault
      3    6   21    0    0    5    0    2       0.00 'fault'
      4   21   32    0    0    5    0    2       0.00 'fault'
* ie. wall blocks
      5   21    6    5    5    5    5    3       0.00 'triangular zone'
      6   21   22    7    6    5    5    3       0.00
      0/
*  ------------------------------------------------------------------------------
* COORDINATE LIST
*
* Cartesian coordinates, positive X right of origin, positive Y up from origin
*
* Allways input coord. #1 so the range default will work as expected.
*
* Use a 0/ coordinate number to end the list
*
* |---------|-----------|-----------|
*  | Coord   |     X     |     Y     |
*  | Number  |   Coord   |   Coord   |
*  |---------|-----------|-----------|
            1     1250.00     1145.00
            5     1250.00     1150.00
            6     1300.00     1150.00
            7     1350.00     1150.00
           10     1350.00     1230.00
           11     1300.00     1245.00
           12     1250.00     1235.00
           21     1270.00     1100.00
           22     1350.00     1100.00
           32     1240.00     1050.00
          101     1200.00     1300.00
          102     1200.00     1050.00
          103     1400.00     1050.00
          104     1400.00     1300.00
            0/
*  ------------------------------------------------------------------------------
* FIELD STRESSES
*
* Material number 1 is reserved for the host material and far field stresses
*
* Theta, is measured counterclockwise from the X-axis to Sigma:1
*
* In vertical models parameter 'a' maybe used to vary field stress with depth.
*
* A total maximum of 10 stress states can be used.
*
* Use a 0/ material number to end the list
```

```
*
*  |---------|-----------------------------------------|---------|
*  |Material |          Principal Stresses             |         |
*  | Number  |Sigma:1 = a * y + b|Sigma:2 = a * y + b  |  Theta  |
*  |---------|----a----|----b----|----a----|----b----|---------|
*  ie. far field stresses in the host material
           1      0.00   5200.00      0.00   3350.00     90.00
*  ie. initial field stresses ore
           2      0.00   5200.00      0.00   3350.00     90.00
*  ie. initial field stresses on fault
           3      0.00   5200.00      0.00   3350.00     90.00
*  ie. initial stresses in backfill material
           4      0.00    100.00      0.00    100.00      0.00
           0/
*  ------------------------------------------------------------------------------
*  MATERIAL PROPERTIES LIST
*
*  Material number 1 is reserved for the host rock (material type 1 or 2)
*
*  For each material the first line defines peak strength properties.
*
*  The second line is reserved for residual strength properties which are not
*  considered in the current version Mine2D but it must be present and contain
*  six dummy values.
*
*  The third line contains the Visco-Plastic parameters G1 & G2 followed by
*  four dummy values in fields reserved for future use.
*
*  Strength Parameter, phi is in degrees
*
*  Material Types
*        0  Elastic  (no Safety Factor calculated)
*       +1  Elasto-Visco-Plastic , Mohr/Coulomb
*       +2  Elasto-Visco-Plastic , Hoek/Brown
*       -1  Visco-Plastic , Mohr/Coulomb
*       -2  Visco-Plastic , Hoek/Brown
*
*  Failure Criteria
*        1 Mohr/Coulomb: Sig:1 = Co + Sig:3.tan(45+phi/2)**2
*                        Sig:N = Co
*                        Sig:S = So + Sig:N.tan(phi)
*        2 Hoek/Brown:   Sig:1 = Sig:3 + ( m.Co.Sig:3 + s.Co**2 )**(1/2)
*
*  Visco-Plasticity parameters
*
*      G1 , G2 (ref. to MINE2D User's Manual for guidelines)
*
*  A total maximum of 10 materials can be used.
*
*  Use a 0/ material number to end the list
*
*  |---------|---- Peak Strength/Residual Strength/G1 - G2 ----------------|
*  |Material |  Young's  |Poisson's |    To    |    Co    |    So,m  |  phi,s   |
*  |Num |Type|Modulus/G1| Ratio/G2 |          |          |          |          |
*  |----|----|----------|----------|----------|----------|----------|----------|
*  ie. host material - rock
       1    1 1.0000E+07       0.30   -300.00   9000.00      0.00     45.00
              0.00            0.00      0.00      0.00      0.00      0.00
              0.8000E+07      0.00      0.00      0.00      0.00      0.00
*  ie. alternate material properties - ore
       2    1 0.7200E+07       0.30   -300.00   8000.00      0.00     35.00
              0.00            0.00      0.00      0.00      0.00      0.00
```

```
            0.3600E+07      0.00      0.00      0.00      0.00      0.00
* ie. properties to be assigned to the fault
*     for a type 2 element Co is a compressive strength cutoff
      3   -1 0.00             0.30      0.00  99999.00      0.00     30.00
            0.00              0.00      0.00      0.00      0.00      0.00
            0.3600E+07        0.00      0.00      0.00      0.00      0.00
* ie. high density cemented backfill
      4    1 2.9000E+04       0.25    -25.00    400.00      0.00     24.00
            0.0               0.00      0.00      0.00      0.00      0.00
            2.9000E+03        0.00      0.00      0.00      0.00      0.00
   0/
* ------------------------------------------------------------------------------
* GRID DEFINITION LIST
*
* Grids are optional, results are always calculated inside type 3 and 4 zones
* when they contain material.  Where grid points are outside such zones results
* will be calculated using the strength criteria of the material indicated.
*
* Use a 0/ grid number to end the list
*
* |-------|---------------------------|-------------|------------|
* |  Grid |      Coordinate Number    | # of Div    |            |
* | Number| Pt.  | Pt.  | Pt.  | Pt.  |Between Pt.  |  Material  |
* |       | One  | Two  |Three | Four | 1-2  | 2-3  |  Number    |
* |-------|------|------|------|------|------|------|------------|
        1    101    102    103    104     20     20      1
       0/
* ------------------------------------------------------------------------------
* MINING STEPS
*
* Code = +n to assign material 'n' to the zone (block)
*      = -n to assign excavation stress state 'n' to the zone
*      =  0 to excavate the zone (block)
*
* Use 0/ zone number to end the list that describes each mining step
*
* See User's Manual for restart instructions
*
* |---------|---------|
* |  Zone   |         |
* | (block) |  Code   |
* |---------|---------|
* Mining Step    1
* special material properties must be assigned in step 1
* assign material 2 properties to the ore zones except zone 5 (type 4)
         1        +2
* assign material 3 properties to the fault
         3        +3
         4        +3
* footwall and hangingwall blocks will have material 1 properties by default
* excavate old opening which will be left unfilled
         2         0
         0/
* ------------------------------------------------------------------------------
```

APPENDIX II

SOFTWARE INSTALLATION NOTES

MINE2D consists of two programs MINE and OUTPUT. This version was prepared using SVS FORTRAN 77 compiler, the Phar Lap 386/DOS-Extender Software Development Kit, and the Phar Lap 386/VMM (virtual memory manager). The programs have been compiled for several problem sizes and for use with either the Intel 387 or Wietek 1167 math coprocessors.

DISK 1 - MINE2D EXECUTABLES

Copy the contents to directory c:\MINE2D

Decompress the files - c:\MINE2D> PKUNZIP MINE2D.ZIP
(2 Mb. disk space required)

MINE	BAT	447	12-17-91	10:13p
MINEVMM	BAT	633	12-17-91	10:39p
OUTPUT	BAT	111	12-17-91	9:40p
MI1200	EXP	164862	11-11-91	5:47p
MI2000	EXP	172386	11-11-91	6:26p
MI300	EXP	160881	11-11-91	10:20p
MI450	EXP	161217	12-08-91	1:43p
MI675	EXP	161964	12-08-91	1:12p
MW1200	EXP	198310	11-10-91	11:27p
MW2000	EXP	205834	10-20-91	5:38p
MW300	EXP	194329	11-11-91	10:31p
MW450	EXP	194665	12-17-91	10:54p
MW675	EXP	195412	12-08-91	1:17p
OUT_I	EXP	218595	12-08-91	5:54p
OUT_W	EXP	243915	12-08-91	5:57p
MINE2D	PAR	340	12-17-91	9:13p
SAMPLE	INP	10691	12-17-91	9:46p
MINE2D	PZD			

Batch files are provided for starting MINE and OUTPUT. Edit the batch files to select the version (size and coprocessor) that you wish to run. See Table 3-1 in the MINE2D USER'S MANUAL.

Copy RUN386.EXE from the PHAR LAP 386/DOS-Extender Development Kit to the MINE2D directory.

OUTPUT requires DEVICE=ANSI.SYS in your CONFIG.SYS file.

If the a virtual memory version of MINE is started with MINEVMM.BAT copy VMMDRV.EXP from the PHAR LAP 386/VMM Virtual Memory Manager to the MINE2D directory. Create a directory C:\MINE2D\SWAP for the scratch files. If the computer is restarted to abort an analysis. Run CHKDSK /F to recover the lost disk space. DO NOT CHAIN THE LOST CLUSTERS.

If PIZAZZ PLUS is installed for printing plots to a Paintjet printer, the MINE2D.PZD settings file used for the manual may be loaded.

DISK 2 - MINE2D EXAMPLE FILES

Copy the files EXAMPLE1.ZIP and EX1TXT.ZIP to the MINE2D directory and decompress the files desired.

ie. PKUNZIP EXAMPLE1

```
EXAMPLE1 001     174180 11-28-91    7:30p
EXAMPLE1 002     174540 11-28-91    8:13p
EXAMPLE1 003     174900 11-28-91    8:53p
EXAMPLE1 004     175260 11-28-91    9:33p
EXAMPLE1 005     175620 11-28-91   11:18p
EXAMPLE1 INP       7320 11-26-91   11:46p
EXAMPLE1 OUT       9412 11-28-91   11:18p
```

ie. PKUNZIP EX1TXT

```
EXAMPLE1 TXT    1104586 11-28-91   11:18p
```

DISK 3 - MINE2D SOURCE CODE

The source code MINE and OUTPUT, and batch files to compile and link the Intel and Wietek versions are contained in compressed files MINECODE.ZIP and OUT_CODE.ZIP. Copy them to seperate directories before decompressing the files.

SVS FORTRAN-77 Release 2.8, 386-486/DOS Extended must be installed. (Silicon Valley Software 415-572-8800). SVS.EXE, and 386LINK.EXE (from 386/DOS-Extender) should be present.

Use the MAKE_I.BAT or MAKE_W.BAT file to compile the code.